GISRED v1.0

WATER DISTRIBUTION MODEL BUILDER EXTENSION

USER'S MANUAL

GISRed 1.0 EXTENSION

by Hugo J. Bartolín

GISRed Water Distribution Model Builder Extension v1.0,
an integration of ArcView GIS 3.x (ESRI) and Epanet v2.0 build 2.00.10 (USEPA).

Directed by Fernando Martínez Alzamora
fmartine@dihma.upv.es
Developed by Hugo J. Bartolín
hbartoli@dihma.upv.es

REDHISP Group
http://www.redhisp.upv.es
Institute of Water and Environmental Engineering
Polytechnic University of Valencia
Valencia, Spain

March, 2004

TABLE OF CONTENTS

WHAT IS THE GISRED EXTENSION? 7

INSTALLATION 8

SYSTEM REQUIREMENTS 9

1.1 HARDWARE 9
1.2 SOFTWARE 9

EXTENSION LOADING 10

STEPS IN USING GISRED 11

ABOUT THIS MANUAL 12

CHAPTER 1. QUICK START TUTORIAL 13

1.1 EXAMPLE NETWORK 13
1.2 PROJECT SETUP 13
1.3 DRAWING THE NETWORK 15
1.4 REPOSITIONING OBJECTS 17
1.5 SETTING PROPERTIES 18
1.5.1 Setting Node Properties 18
1.5.2 Setting Link Properties 19
1.6 QUERYING THE NETWORK 20
1.7 ADDING A PUMP CURVE 21
1.8 SAVING AND OPENING PROJECTS 22
1.9 USING THE GISRED PROJECT MANAGER 23
1.10 RUNNING A SINGLE PERIOD ANALYSIS 24
1.11 ADDING A TIME PATTERN 25
1.12 CREATING THE TIME PATTERN 26
1.13 RUNNING AN EXTENDED PERIOD SIMULATION 28
1.14 OTHER THINGS 28

CHAPTER 2. GISRED'S GUI (GRAPHICAL USER INTERFACE) 29

2.1 EXTENSION WORKSPACE 29
2.1.1 PROJECT GUI (ArcView Project Manager) 30
2.1.2 SCENARIOS GUI (GISRed environment) 31

CHAPTER 3. WORKING WITH GISRED PROJECTS 37

3.1 GISRED PROJECT MANAGER 37
3.2 OPENING AND SAVING PROJECTS 39
3.3 PROJECT DEFAULTS 40
3.4 SCENARIO PROPERTIES 42

CHAPTER 4. NETWORK ELEMENTS 43

4.1 TYPE OF NETWORK ELEMENTS 43
4.2 EDITING TOOLS 43
4.3 ADDING NETWORK ELEMENTS 46
4.3.1 Adding a Link 46
4.3.2 Adding a Node 47
4.4 SELECTING ELEMENTS 48
4.5 DELETING AN ELEMENT 51
4.6 MOVING A JUNCTION 52
4.7 OTHER OPERATIONS 53
4.7.1 Modifying Pipe Layout 53
4.7.2 Connecting Pipes 53
4.7.3 Creating a T Connection 54

4.8 CREATING AN IMAGE CATALOG IN GISRED 55
4.8.1 ADDING AN IMAGE TO AN IMAGE CATALOG 57
4.8.2 DELETING AN IMAGE FROM AN IMAGE CATALOG 57
4.9 OVERVIEW MAP 58

CHAPTER 5. IMPORTING A NETWORK TO THE SCENARIO 59

5.1 CAD 60
5.1.1 CAD DRAWING COORDINATE TRANSFORMATIONS 62
5.2 SHAPEFILES 64
5.3 ERROR THEMES AND NAVIGATOR 66
5.4 GIS COVERAGES 67
5.5 EPANET INPUT FILES 67
5.6 GISRED PROJECT 67

CHAPTER 6. EDITING NETWORK PROPERTIES 69

6.1 JUNCTION PROPERTIES 70
6.1.1 Emitters 72
6.1.2 Multiple Demand Editor 72
6.2 RESERVOIR PROPERTIES 73
6.3 TANK PROPERTIES 75
6.3.1 Mixing in Storage Tanks 77
6.4 PIPE PROPERTIES 79
6.4.1 Water Quality Reactions 81
6.5 PUMP PROPERTIES 84
6.6 VALVE PROPERTIES 86
6.6.1 Type of Valves 88
6.7 EDITING A GROUP OF NETWORK ELEMENTS 90

CHAPTER 7. PATTERNS 91

7.1 GISRED PATTERN EDITOR 91

CHAPTER 8. CURVES 95

8.1 GISRED CURVE EDITOR 96

CHAPTER 9. CONTROLS 99

9.1 SIMPLE CONTROLS 99
9.1.1 GISRed Simple Controls EDITOR 100
9.2 RULE-BASED CONTROLS 104
9.2.1 GISRed Rule-Based Controls EDITOR 108

CHAPTER 10. GISRED TOC THEME MANAGER 109

10.1 TOC (TABLE OF CONTENTS) 109
10.2 TOC THEME MANAGER 110
10.3 GISRED CUSTOM-BUILT THEMES 111

CHAPTER 11. QUERYING THE NETWORK 113

CHAPTER 12. NETWORK TOOLS 115

12.1 CHECKING TOOLS 115
12.1.1 Verification Options 115
12.2 CONNECTIVITY TOOLS 116

CHAPTER 13. ELEVATION INTERPOLATION 119

CHAPTER 14. DEMAND ALLOCATION 123

CHAPTER 15. ANALYSIS OPTIONS 127

15.1 SETTING ANALYSIS OPTIONS 127
15.1.1 Hydraulic Options 128
15.1.2 Water Quality Options 129

15.1.3 Reaction Options 130
15.1.4 Times Options 131
15.1.5 Energy Options 132

CHAPTER 16. NETWORK SIMULATION 133

16.1 EXPORTING EPANET INPUT FILES 133
16.2 RUNNING AN ANALYSIS 133
16.3 VIEWING RESULTS ON THE SCENARIO 134

CHAPTER 17. THE NETWORK MODEL CALIBRATION 137

17.1 INTRODUCTION 137
17.2 NETWORK PROPERTIES 138
17.3 CALIBRATION PROCESS 142
17.4 GROUPING THE NETWORK 148
17.5 CALIBRATION INPUT FILES 149
17.6 RUNNING A CALIBRATION AND IMPORTING RESULTS 149
17.7 VIEWING CALIBRATION RESULTS 150
17.7.1 STATISTICS (Refer to Z. Kapelan thesis, 2002) 153

APPENDIX A. THE STANDARD GA LIBRARY 157

A.1 GENERAL GENETIC ALGORITHM CONCEPT 157
A.2 GENETIC ALGORITHM TYPES 158
A.2.1 Steady State GA 158
A.2.2 Generational GA 158
A.3 SELECTION TECHNIQUES 158
A.3.1 Uniform Random 159
A.3.2 Rank Biased 159
A.3.3 Roulette Wheel 159
A.3.4 Tournament 159
A.3.5 SRSWR (Stochastic Remainder Selection Without Replacement) 160
A.4 CROSSOVER TECHNIQUES 161
A.4.1 None 161
A.4.2 Randomise 161
A.4.3 Simple One Point Crossover 161
A.4.4 Uniform Random 161
A.4.5 Arithmetic 162
A.5 MUTATION TECHNIQUES 162
A.5.1 Uniform Random 163
A.5.2 Uniform Random by Gene 163
A.5.3 Non-Uniform Random 163
A.6 CONVERGENCE CRITERIA 164
A.7 OTHER VARIABLES 164
A.7.1 Population Size 164
A.7.2 Random Number Seed 164
A.7.3 Output Interval 164

APPENDIX B – UNITS OF MEASUREMENT 165

REFERENCES 167

TABLE OF ILLUSTRATIONS

Fig. 1 Loading GISRed Extensión 10
Fig. 2 Scenarios Document 10
Fig. 3 GISRed Model Building Process Flowchart. 11
Fig. 4 Quick Tutorial. Example Network. 13
Fig. 5 Quick Tutorial. New Project Dialog. 14
Fig. 6 Quick Tutorial. Defaults Dialog. 14
Fig. 7 Quick Tutorial. Pipe Editing Tools. 15
Fig. 8 Quick Tutorial. Node Editing Tools. 16
Fig. 9 Quick Tutorial. Editing pipe vertices. 17
Fig. 10 Quick Tutorial. Junction Properties. 19
Fig. 11 Quick Tutorial. Pipe Properties. 20
Fig. 12 Quick Tutorial. Network Queries. 20
Fig. 13 Quick Tutorial. Pump Properties. 21
Fig. 14 Quick Tutorial. Adding a Pump Curve. 21
Fig. 15 Quick Tutorial. Using the GISRed Project Manager. 23
Fig. 16 Quick Tutorial. Browsing Results. 24
Fig. 17 Quick Tutorial. Analysis Options Dialog. 25
Fig. 18 Quick Tutorial. Adding a New Demand Pattern. 26
Fig. 19 Quick Tutorial. Demand Pattern Dialog. Viewing a Pattern Chart. 26
Fig. 20 Quick Tutorial. Group Editing. 27
Fig. 21 GISRed Workspace. 29
Fig. 22 New GISRed Project Dialog. 30
Fig. 23 GISRed Project Manager. 31
Fig. 24 Scenario. 36
Fig. 25 GISRed Project Manager Dialog. 37
Fig. 26 Creating a New GISRed Project. 38
Fig. 27 GISRed Projects Defaults. 40
Fig. 28 Scenario Properties. 42
Fig. 29 Scenario Properties. Network Details. 42
Fig. 30 ArcView Query Builder Dialog. 48
Fig. 31 Selecting features using graphics. 50
Fig. 32 Moving a Node. 52
Fig. 33 Modifying Pipe Layout. 53
Fig. 34 Connecting Pipes. 54
Fig. 35 Creating a T connection. 54
Fig. 36 Adding an image catalog to the scenario. 55
Fig. 37 First image to the Catalog. Tic Selection Dialog. 56
Fig. 38 Image Catalog Theme. 56
Fig. 39 Selecting the type of data to import. 59
Fig. 40 CAD Importation Dialog. 61
Fig. 41 Applying a Coordinate Transformation. 63
Fig. 42 Shapefile Importation Dialog. 64
Fig. 43 Error Navigator. 66
Fig. 44 Importing an EPANET input file. 67
Fig. 45 Junction Properties. 70
Fig. 46 Reservoir Properties. 73
Fig. 47 Tank Properties. 75
Fig. 48 Pipe Properties. 79
Fig. 49 Reactions zones within a pipe. 81
Fig. 50 Pump Properties. 84
Fig. 51 Valve Properties. 86
Fig. 52 Group Edit Dialog. 90
Fig. 53 GISRed Pattern Editor. 92
Fig. 54 GISRed Curve Editor. 97

Fig. 55 Control Rules Dialog. 99
Fig. 56 GISRed Simple Controls Editor. 100
Fig. 57 Adding a new simple control. 101
Fig. 58 Report of Simple Controls Imported. 103
Fig. 59 Rule-based Controls Editor. 108
Fig. 60 GISRed TOC Theme Manager. 110
Fig. 61 Adding a new GISRed Theme. 112
Fig. 62 Network Queries Dialog. 113
Fig. 63 Network Queries. Diameters and Base Demands. 113
Fig. 64 Network Checking Options. 115
Fig. 65 Connectivity Dialog. 116
Fig. 66 Checking the network connectivity. Number of Sub-networks. 117
Fig. 67 Network tree and co-tree. 118
Fig. 68 Elevation Interpolation. Theme of Elevations. 120
Fig. 69 Elevation Interpolation. Interpolation Surface. 121
Fig. 70 Elevation Interpolation Dialog. 122
Fig. 71 Demand Allocation. Consumption Table. 123
Fig. 72 Demand Allocation Dialog. 124
Fig. 73 Demand Allocation. Node-Road Segment Relation Theme. 124
Fig. 74 Demand Allocation. Report Window. 125
Fig. 75 Analysis Options Dialog. 127
Fig. 76 Epanet Report Window. 133
Fig. 77 Viewing Results on the Scenario. 134
Fig. 78 Anytown Network Layout. 137
Fig. 79 Calibration Example. Demand Pattern. 141
Fig. 80 GISRed Calibration Process. Flowchart. 142
Fig. 81 New Calibration Configuration. 143
Fig. 82 Loading Conditions Dialog. 144
Fig. 83 Importing Field Data. 144
Fig. 84 Field Data after the importation. 145
Fig. 85 GA Parameters dialog. 146
Fig. 86 Calibration Manager. 147
Fig. 87 Calibration menu bar options. 147
Fig. 88 Group Editing Dialog. 148
Fig. 89 Pipe Property Dialog. Changing Calibration Roughness Groups. 149
Fig. 90 Calibration Results Dialog. 151
Fig. 91 Different types of graphs showing calibration results. 152

WHAT IS THE GISRED EXTENSION?

GISRed Extension is an Extension to ESRI's ArcView GIS 3.2 software that integrates the widely used hydraulic modelling software EPANET 2.0 within ArcView GIS 3.x, keeping all the original GIS options.

Consequently, this 'add-in' application may be used to perform simple tasks such as drawing a basic network model from scratch or much more complex such as importing a whole dataset from an external source, creating an integrated database, building a network model and calibrating it.

The ArcView GISRed Extension is essentially a tool for helping the hydraulic engineer in the task of water distribution network modelling and decision-making process within a GIS environment.

With the GISRed Extension, the user will be able to perform the following functions found on the extension menus:

- Management of multiple GISRed projects using the customized Project Manager. The portability of the projects is assured using the extension.
- Multiple copies (clones) of the same project.
- Data Import from different sources such as CAD files, EPANET files, ArcInfo coverages…
- Interpolation of elevations at each node.
- Demand allocation at selected nodes.
- Network calibration using a Genetic Algorithm search method.
- Definition of as many calibration configurations as required.
- Running extended period simulations and retrieving the results showing them on the scenario.
- Network queries and see the results in a specific featured theme.
- Checking the connectivity of the network based upon graph theory algorithms.

INSTALLATION

Where to download GISRed Extension

A trial version of GISRed Extension can be downloaded from the REDHISP Group web page. The URL is: http://www.redhisp.upv.es/software/GISRed

An executable file is available ready to be installed. The installation process is very straightforward and takes just a few seconds.

In order to install the extension, run the executable file following the entire installer wizard. When the installer prompts the user to select the installation path, the user should enter ArcView's EXT32 folder path. By default this path is: c:\ESRI\av_gis30\arcview\EXT32.

It is possible to make a customized installation, selecting just those components needed (examples, tutorials…). By default, all components will be installed.

If the full installation is chosen, the installation program places a file called *GISRed.avx* and a folder called 'GISRED' on your system in the $AVEXT directory. All the files and folders installed with the extension are specified below:

Note: AVEXT is an environment variable set in ArcView's startup script that references the default ArcView extension directory.

PATH	DESCRIPTION
$AVEXT/GISRED.avx	Extension
$AVEXT/GISRED/Calibrator	Calibrator directory
$AVEXT/GISRED/Calibrator/Calibrator.exe	Calibrator module
$AVEXT/ GISRED /Calibrator/epanet2.dll	Epanet 2 dynamic link library
$AVEXT/GISRed/Examples	Examples directory
$AVEXT/GISRED/Examples/ANYTOWN	Anytown Calibration Example.
$AVEXT/GISRED/Examples/NET1	GISRed Net 1 Example
$AVEXT/GISRED/Examples/NET2	GISRed Net 2 Example
$AVEXT/GISRED/Examples/NET3	GISRed Net 3 Example
$AVEXT/GISRED/Examples/Tutorial	GISRed Tutorial. Contains the network used in the tutorial.
$AVEXT/GISRED/Icons	Icons directory. Icons used by the application
$AVEXT/GISRED/Manual	Manual directory
$AVEXT/GISRED/Manual/GISRedTUTORIAL.htm	GISRed Tutorial in HTML format
$AVEXT/GISRED/Manual/GISRedTUTORIAL	GISRed Tutorial folder. HTML references. Needed by the htm file.
$AVEXT/GISRED/Manual/Calibration.htm	GISRed Calibration Tutorial in HTML file.
$AVEXT/GISRED/Manual/Calibration	GISRed Calibration Tutorial folder. HTML references needed by the htm file.
$AVEXT/GISRED/Tablas	Table directory. Tables needed by the application.
$FONT/GISRED.ttf	Font loaded and installed by the GISRed installer.

SYSTEM REQUIREMENTS

1.1 HARDWARE

MINIMUM

COMPUTER: Industry-standard personal computer with at least a Pentium or higher Intel-based microprocessor.

HARD DISK: 15 MB

RAM Memory: 32 MB

RECOMMENDED

COMPUTER: Pentium III or higher

RAM Memory: 64 MB

1.2 SOFTWARE

Windows 98/ ME, Windows NT 4.0, 2000, Windows XP--Home Edition and Professional (supported with limitations).

ArcView GIS 3.2

ArcView Spatial Analyst Extension

Epanet 2.00 to run GISRed-generated exported input files.

EXTENSION LOADING

Use the ArcView Extensions dialog to load and unload the GISRed Extension. GISRed extends ArcView "on the fly" enhancing your working environment with additional hydraulic modelling-oriented objects, dialogs and menus independent of the current project.

➢ To **load the Extension**:

1. Start ArcView GIS.
2. Open the Extensions Dialog (**File | Extensions**).

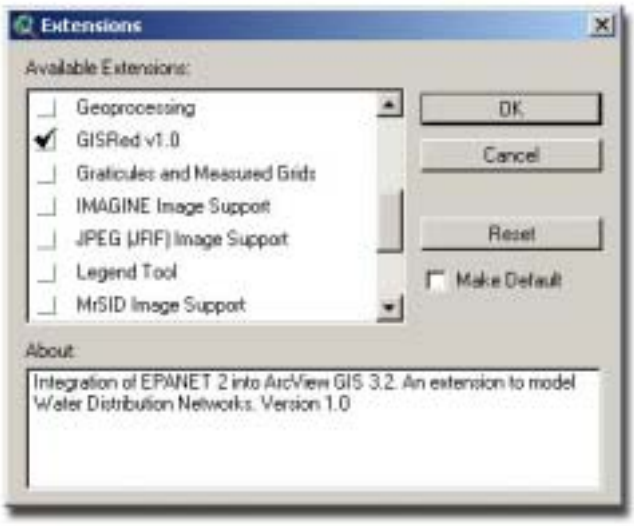


Fig. 1 Loading GISRed Extensión

3. Select the GISRed Extension (switch on the checkbox on the left side).
4. Once the extension is loaded, a new document called 'Scenarios' appears in the ArcView Project Manager Window.

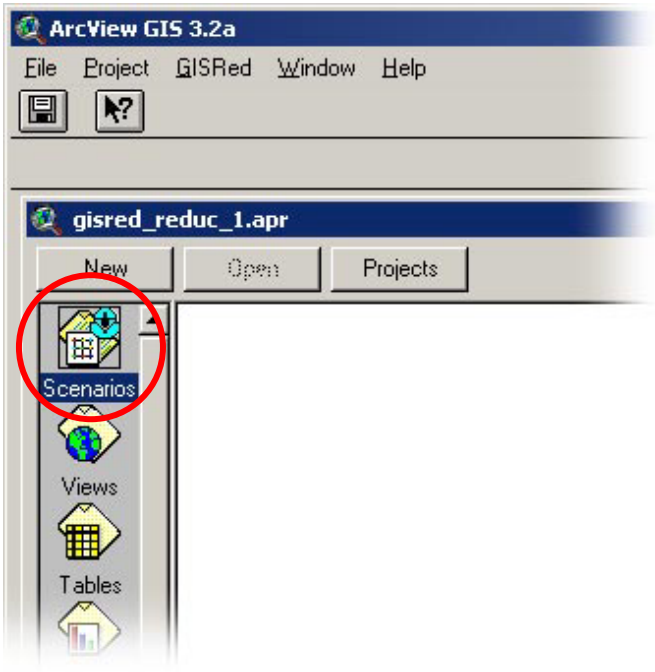


Fig. 2 Scenarios Document

STEPS IN USING GISRED

The typical steps carried out when using GISRed Extension to build a water distribution model are:

1. Draw a network representation of your distribution system from scratch using the extension tools (see 4.2) or import a basic description of the network placed in a CAD file, a shapefile or Epanet input file (see CHAPTER 5).
2. Check importation errors in case of importing (see 5.3)
3. Simplify the network (if required).
4. Edit the properties of the objects that make up the system (see CHAPTER 6).
5. Describe how the system is operated by means of demand patterns, pump curves, control rules, etc. (see CHAPTER 7, CHAPTER 8 and CHAPTER 9).
6. Interpolate elevations at selected nodes (see CHAPTER 13).
7. Allocate demands (see CHAPTER 14).
8. Select a set of analysis options (see CHAPTER 15).
9. Calibrate the network manually as a first approach.
10. Run a hydraulic/water quality analysis (see 16.2).
11. View the results of the analysis (see 16.3).
12. Define a calibration configuration (see CHAPTER 17).
13. Calibrate the network using the GISRed GA calibrator.
14. Commit the calibration results.
15. Perform a new simulation.
16. Analyze the results again.

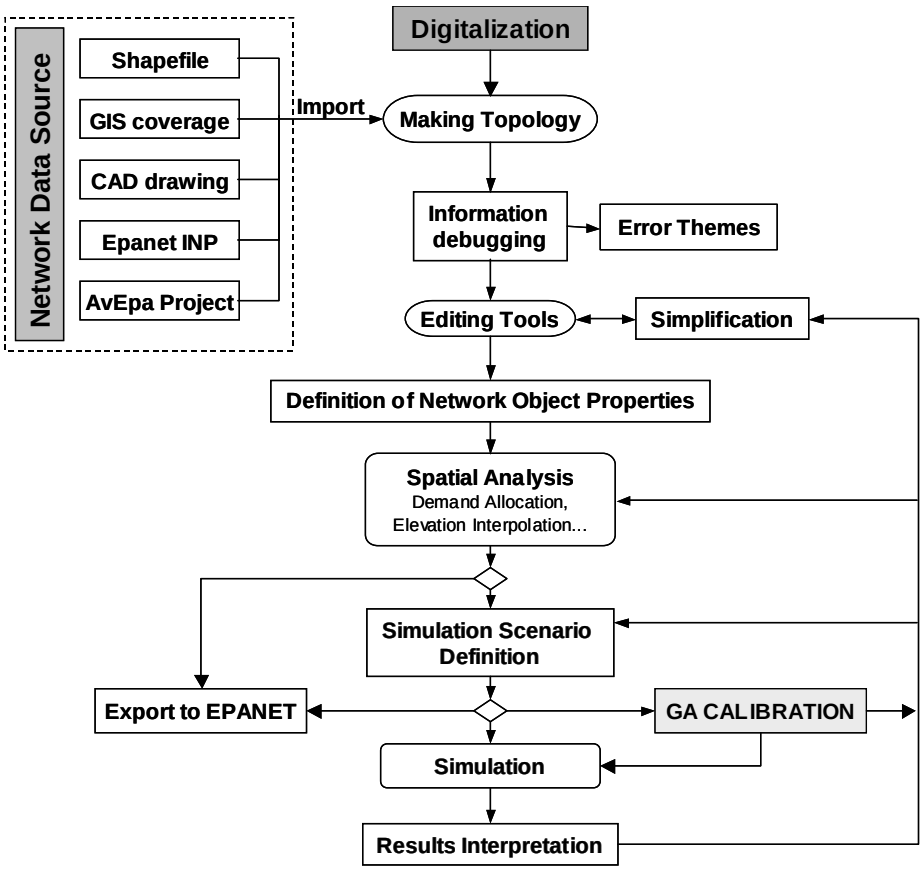


Fig. 3 GISRed Model Building Process Flowchart.

ABOUT THIS MANUAL

Chapter 1 of this manual offers up a quick tutorial on the use of GISRed extension. It is highly recommended that readers unfamiliar with the basics of ArcView review the ArcView quick start guide before working through this tutorial.

Chapter 2 provides information on GISRed's graphical user interface and how its workspace is organised. The user can find all the menus, buttons and tools associated with the extension. It also explains the concept of scenario as the base of any GISRed water distribution model.

Chapter 3 explains how to work with an GISRed project and the project files that store all of the information contained in an GISRed model of a distribution system. It shows how to create, open, and save these files as well as how to set default project options.

Chapter 4 describes how one goes about building a network model of a distribution system with GISRed. It shows how to create the various physical objects (pipes, pumps, valves, junctions, tanks, etc.) that make up a system, how to edit the properties of these objects, how to select, add or delete those objects and some other useful additional operations when building a model. At the end of this chapter the user can find out how to work with images and catalog images.

Chapter 5 explains how to import a network into the scenario. All formats that can be imported from GISRed are described. The way to import a GISRed project is also explained.

Chapter 6 shows how to edit network properties for each one of the objects. All the options of the dialogs are described. The edition of a group of elements is also considered in this chapter.

Chapter 7 and 8 introduce the GISRed pattern editor and the GISRed curve editor respectively.

Chapter 9 shows the control rules editors. Simple and rule-based controls are explained.

Chapter 10 gives details on how to manage shapefile themes in the scenario. It shows the TOC manager and the custom-built themes.

Chapter 11 focuses attention on how to query the network, that is, to make new themes classified by diameter, demand, elevation, etc.

Chapter 12 shows the tools available to check the connectivity and verify other properties of the network.

Chapter 13 explains how to interpolate elevations in the network nodes by means of a theme of scattered points with an elevation reference.

Chapter 14 is dedicated to the process of demand allocation, the method used and the tools available.

Chapter 15 describes the various options that control how a hydraulic/water quality analysis is made.

Chapter 16 shows how to run a network simulation and how to view the results on the scenario.

Chapter 17 is all about the network model calibration using genetic algorithms. This chapter gives an explanation on the calibration process, and how to interpret the calibration results.

Appendix A shows the particularities of the GA library used as calibrator and Appendix B provides a table of units of expression for all design and computed parameters.

CHAPTER 1. QUICK START TUTORIAL

This chapter provides a tutorial on how to use GISRed Water Distribution Model Builder Extension by means of a quick example. If you are not familiar with the environment and options of ArcView GIS, you should review the ArcView GIS User's Manual.

1.1 EXAMPLE NETWORK

In this tutorial we will analyze the simple distribution network shown below. It consists of a source reservoir (e.g., a treatment plant clearwell) from which water is pumped into a two-loop pipe network. There is also a pipe leading to a storage tank that floats on the system.

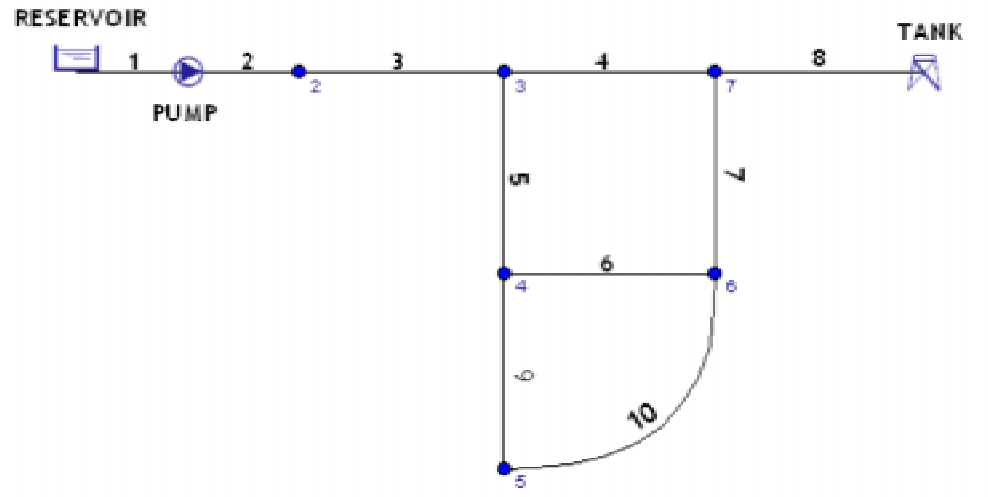


Fig. 4 Quick Tutorial**.** Example Network.

1.2 PROJECT SETUP

Our first task is to create a new project in GISRed and make sure that certain default options are selected.
As mentioned before, GISRed is an ArcView extension, thus if ArcView is not already running then launch it from the Windows Start menu.

1. Once opened, go to **Extensions** in the **File** menu when the ArcView Project Window is active and select **GISRED**. Pressing the OK button will load the extension, and things will be ready to begin.

2. Select **GISRed** | **New** to create a new project. Alternatively the button **New** on the **Scenarios** Document of the ArcView Project Window can be used to create a new GISRed project.

3. Enter a Name for the Project, a path for the database to be stored and a name for the parent scenario. These properties are necessary to create the new project. Finally, press **Create**.

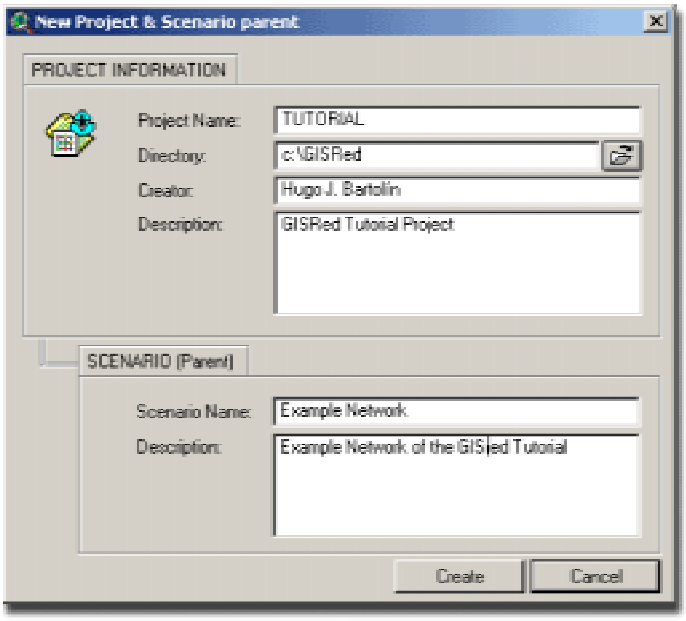


Fig. 5 Quick Tutorial. New Project Dialog.

A new empty scenario is created and two basic themes are added. [Note that a new directory is also created with all the necessary tables to build a water network model. In this example the new directory corresponds to c:\GISRed\Tutorial]

1. Now, select **Scenario** | **Defaults** to open the **Project Defaults** dialog form.

2. On the **ID Labels** page, enter all of the ID Prefix fields or leave the label prefixes by default.

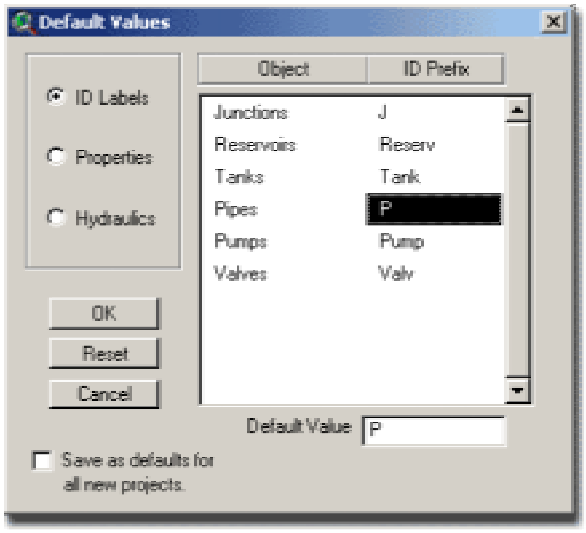


Fig. 6 Quick Tutorial. Defaults Dialog.

3. On the **Hydraulics** page of the dialog choose **LPS** as *Flow Units* and **Hazen-Williams** (H-W) as *Headloss Formula*.

4. Click **OK** to accept the changes and close the dialog.

If you wanted to save these changes for all future new projects you could check the *"Save as defaults…"* checkbox at the bottom of the form before accepting it.

1.3 DRAWING THE NETWORK

We are now ready to begin constructing our network.

Firstly, we will add the pipes.

Fig. 7 Quick Tutorial. Pipe Editing Tools.

Let's begin with Pipe 1 connecting the Reservoir to Node 2. [The Pipe is first drawn, then the Reservoir and Pump will be added later on].

1. Make the Link Theme Active.
2. Go to the **Edit Model** | **Editing Tools** menu or alternatively, use the button on the toolbar or the pop-up menu.
3. Click the first button on the left .
4. Click the mouse to enter the first point of the pipe and double-click when entering the last point to tell GISRed that you are finished entering vertices.
5. Note how an outline of the pipe is drawn as you move the mouse from the reservoir to node 2.
6. Note also how two new nodes appear at the ends of the pipe.

7. Repeat this procedure for pipes 2 through 9. *[The Tank will also be added later on]*

   Every time you draw a pipe clicking very close to an existing node (within a tolerance), the pipes will snap. This way, all pipes coming together at an intersection will share the same node, since the snapping environment is automatically set. The connectivity degree of the node will be equal to the number of pipes intersecting at that point.

8. Pipe 10 is curved. To draw it, click the mouse first on Node 5. Then as you move the mouse towards Node 6, click at those points where a change of direction is needed to maintain the desired shape. Complete the process by double clicking on Node 6.

At this point we will add the rest of the objects.

Fig. 8 Quick Tutorial. Node Editing Tools.

If the link toolbar is still open, it is enough to make the node theme active in order to see the node tool bar, otherwise, make the node theme the active one and use the same button/menu as for links to open the node tool bar.

1. Add the reservoir by clicking the reservoir button on the Node Toolbar. Then click on the node firstly drawn. *[Notice how the cursor icon changes]*

2. Add the tank by clicking the tank button and then clicking on the node where the tank is located at in Fig. 4. *[Notice how the cursor icon changes]*

3. Finally add the pump by pressing the tool with the pump icon, and clicking on the middle of pipe 1. The pipe is split and the pump is inserted.

1.4 REPOSITIONING OBJECTS

At this point we have completed drawing the example network. Your network should look like the one seen in Fig. 4. If the nodes are out of position you can move them around by using the tool , clicking the node to select it, and then clicking again to its new position. The pipes connected to that node will move along with it.

➢ To reshape any of the pipes, use the tool of the link toolbar:

1. First click on the pipe to select it. Automatically a new red outline shows up.
2. New vertices can be added to the outline just by clicking on it.
3. To move a vertex, place the pointer on it (the cursor will change) and then drag it with the left mouse button held down to its new position.
4. To delete a vertex, place the pointer on it and use the *'Del'* key.
5. Click anywhere on the scenario to accept the new shape.

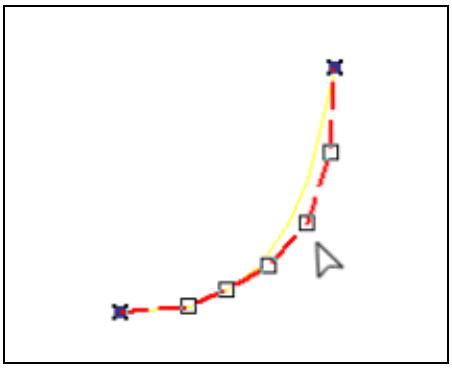


Fig. 9 Quick Tutorial. Editing pipe vertices.

1.5 SETTING PROPERTIES

As objects are added to our scenario, GISRed assigns them a default set of properties. To change the value of a specific property for an object we must select the object while the tool is selected. Depending on the selected element the corresponding property editor dialog is opened.

1.5.1 Setting Node Properties

The nodes in our example network are assumed to have the following properties:

Node	Elevation (m)	Demand (l/s)
RESERVOIR	200	0
2	210	0
3	220	9.5
4	210	9.5
5	200	12.6
6	210	9.5
7	210	0
TANK	253	0
PUMP (Suction)	210	
PUMP (Discharge)	210	

1. Let us begin by selecting Junction 2 into the Property Dialog. Keeping active the node theme in the table of contents (TOC), click on it.

2. Enter the elevation and demand for this node in the appropriate gaps. After this, we need only click on another node to have its properties appear next in the Property Editor Dialog. (We could also press the top dialog buttons to move to the next or previous object of the same type in the database). Thus we can simply move from one junction to the next and fill in our elevations and demands.

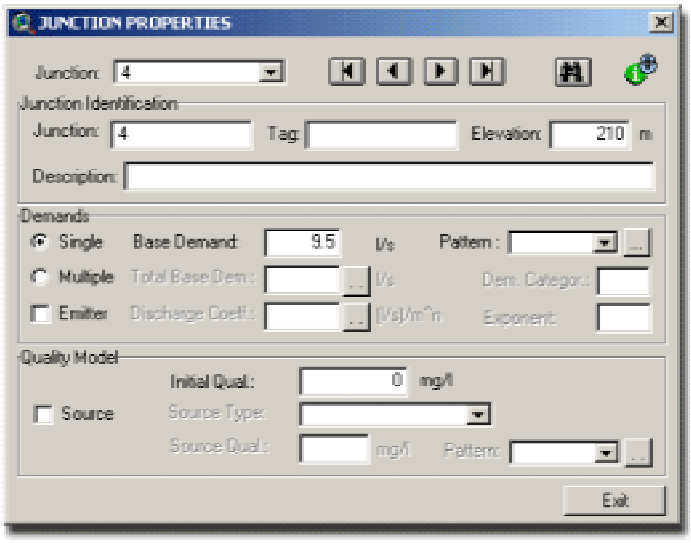


Fig. 10 Quick Tutorial. Junction Properties.

3. For the Reservoir, enter its elevation of 210 m in the *Total Head* field. For the tank, enter 253 m for its elevation, 2 m for its *Initial Level*, 6 for its *Maximum Level*, and 18 m for its *Diameter*.

1.5.2 Setting Link Properties

Assume that the pipes in our network have the following lengths and diameters:

Pipe	Length (m)	Diameter (mm)
1	1	400
2	1	400
3	1000	350
4	1500	300
5	1500	200
6	1500	200
7	1500	200
8	2000	250
9	1500	150
10	2000	150

and that all Roughness Coefficients (C-Factors) are 100 except for pipes 1 and 2 (see Fig. 4) which is 140.

Following the same procedure used for nodes, we simply click on each pipe (keeping the link theme active) to move from pipe to pipe to enter its properties into the Property Editor (it is also possible to use the top browser of the dialog).

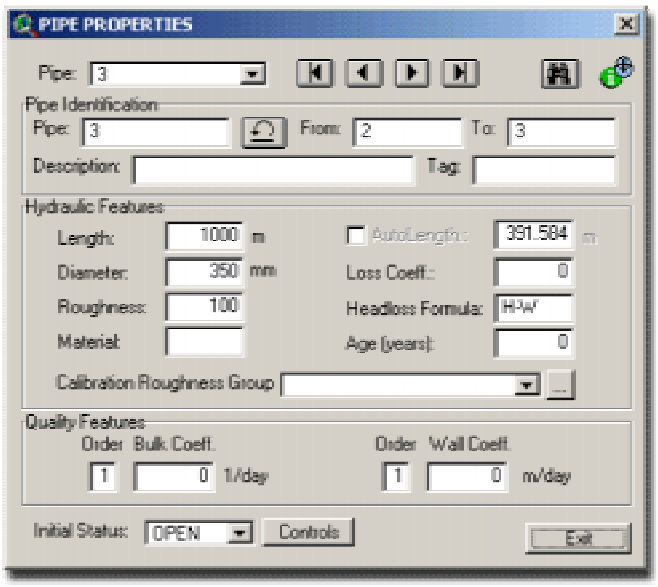


Fig. 11 Quick Tutorial. Pipe Properties.

1.6 QUERYING THE NETWORK

Now that we have entered all the properties of the elements of the network, it is time to check all those properties visually. To do that, there is a special dialog in GISRed called '**Network Queries**'.

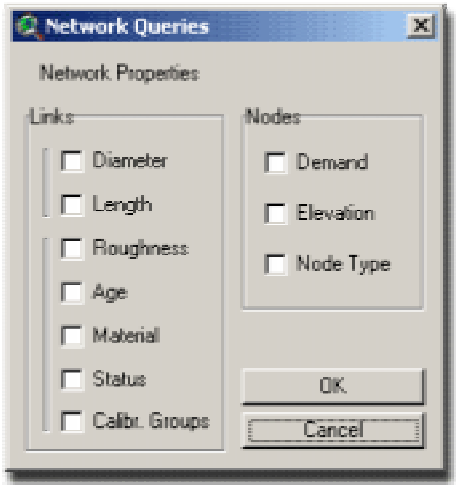


Fig. 12 Quick Tutorial. Network Queries.

Let us check the diameters for instance:

1. Select **Model Tools** | **Queries…** in the menu bar or press .
2. Check the **Diameter** (*inner pipe diameter*) option.
3. Press OK

A new theme classified by diameters is added to the scenario. Try other properties in the same dialog by yourself.

1.7 ADDING A PUMP CURVE

1. For the pump, we need to assign it a pump curve (head versus flow relationship).

2. Select the pump (keeping the node theme active and having pressed the "model properties" tool beforehand) to open the dialog. Since no curves have been entered, press the button beside the ***Pump Curve*** dropdown list to open the curve editor and enter the new curve.

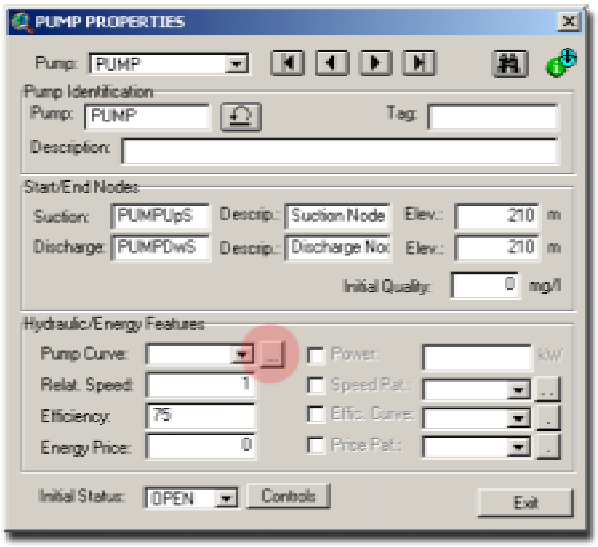


Fig. 13 Quick Tutorial. Pump Properties.

3. Once opened, select **New** Curve and enter a name for the curve, '*Pump1'* is the default name. Enter then the pump's design flow (38 lps) and head (45 m) into the table form. GISRed will automatically create a complete pump curve from this single point. The curve's equation is shown along with its shape is the **Show** button is pressed. Accept the curve to go back to the initial dialog.

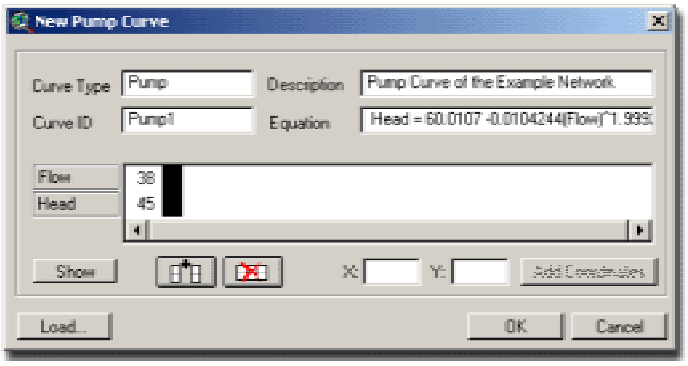


Fig. 14 Quick Tutorial. Adding a Pump Curve.

4. Press return to pass the curve to the pump property editor dialog. The *Pump1* curve will appear in the dropdown list.

1.8 SAVING AND OPENING PROJECTS

Even though it is not strictly necessary to save the GISRed project, since any change in our model is automatically committed to the database, it is possible to save the entire ArcView session (including any other documents opened such as layouts, tables, scripts, etc..) to a file at this point. The file will have the typical *.apr ArcView extension, and will be dependant upon the GISRed extension.

1. From the **File** menu select the **Save Project As** option.

2. In the **Save Project As** dialog that appears, select a folder and file name under which to save this project. We suggest naming the file *tutorial.apr*. (An extension of .apr will be added to the file name if one is not supplied).

3. Click **OK** to save the project to file.

The project data is saved to the file in a text format. To open our project at some later time, we would select the ***Open Project…*** option while the ArcView Project Manager Window is active.

ALTERNATIVELY, *it is possible to exit the session without saving the ArcView Project. The GISRed Project Manager enables the user to open, remove, add or clone GISRed Projects from a list of existing projects already created. The only difference in relation to an ArcView project is that the GISRed project does not keep information of any other documents different from the Scenario itself. Thus, if you are interested in keeping in the project all document objects such as layouts, tables, scripts, etc, you should choose to save the entire project using the first way described.*

1.9 USING THE GISRED PROJECT MANAGER

The GISRed project manager comprises a set of buttons that allow the user to make **new** projects, to **open** existing ones from the list, to **add** a new project to the list (*portability*), **remove** a project from the list, make a **clone** of a project (an exact copy of it) and see / change some properties.

Fig. 15 Quick Tutorial. Using the GISRed Project Manager.

To open our example in a further session:

1. Press the button **Projects** from the **ArcView Project Manager** (*Scenarios*). Or alternatively, go to **Network** | **Project Manager** on the menu bar. The project manager is opened then.

2. Select the **TUTORIAL** example from the list of projects and press Open.

3. The Scenario is opened and the TOC appears just as it was in the last session.

4. Now the scenario is ready to be used again.

5. Exit the Project Manager.

1.10 RUNNING A SINGLE PERIOD ANALYSIS

We now have enough information to run a single period (or snapshot) hydraulic analysis on our example network. To run the analysis let us take the two next steps:

1. Generate an **EPANET Input File**. Select **Scenario** | **Export Epanet Input File** (or click the button).

2. Once the File has been successfully created, select **Scenario** | **Run Analysis** and choose the file that has just been made.

    a. A **Status Report** window will appear with a summary of the process or if the run was unsuccessful then indicating what the problem was

    b. If it ran successfully, computed results will be available from the result **browser** that comes up after the running.

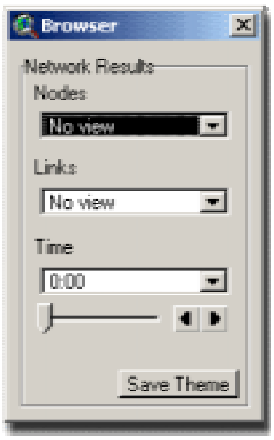


Fig. 16 Quick Tutorial. Browsing Results.

3. Select **Node Pressure** from the Browser's dialog and observe how pressure values at the nodes become color-coded. To edit the Legend for the color-coding, select **Theme** | **Edit Legend**… (alternatively, click or double click on the theme legend). To change the legend intervals, just change the **Value** and **Label** Fields. To change the colors, double click on the symbol and use the **Palette Editor**.

4. See a tabular listing of results by selecting **Theme** | **Table**… (or by clicking the button).

1.11 ADDING A TIME PATTERN

To make our network more realistic for analyzing an extended period of operation we will create a **Time Pattern** that makes demands at the nodes vary in a periodic way over the course of a day. For this simple example we will use a pattern time step of 6 hours. This will cause demands to change at four different times of the day. (A 1-hour pattern time step is a more typical number and is the default assigned to new projects). To set the pattern time step as well as the simulation duration:

1. Select **Scenario** | **Analysis Options** | **Times-Energy** from the menu bar and dialog respectively.

2. Enter 6 for the value of the **Pattern Time Step**.

3. Enter 72 hours (3 days) for the simulation **Duration**.

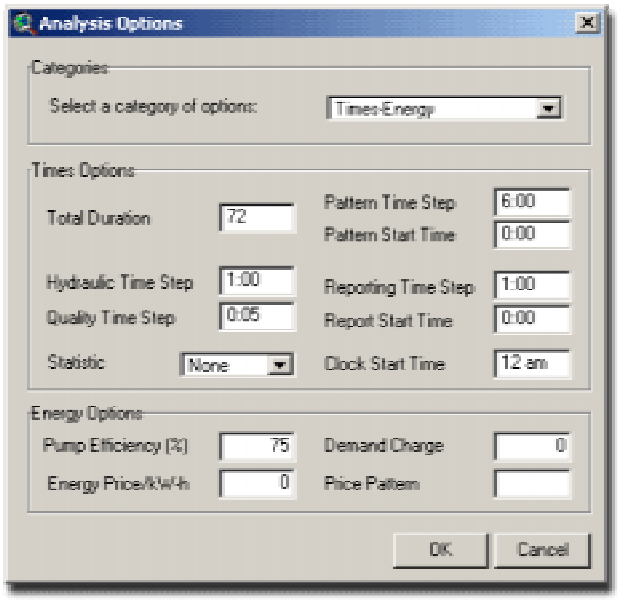


Fig. 17 Quick Tutorial. Analysis Options Dialog.

1.12 CREATING THE TIME PATTERN

To create the time pattern:

1. Select the **Patterns** option in the **Edit** Model menu.

2. Select **Demand** category and click on the **New** button. By default, the new pattern is named *Demand1* and is ready to be edited.

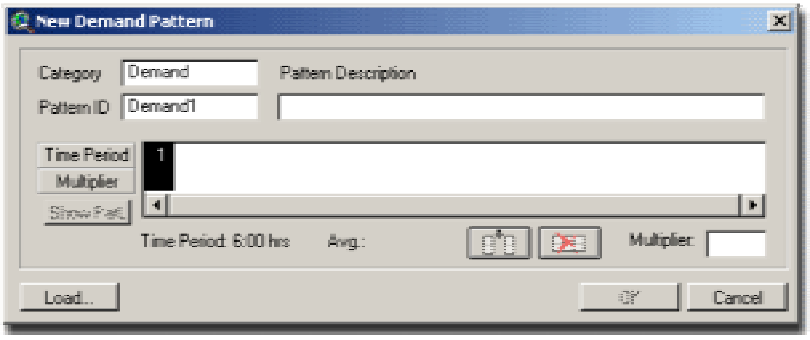
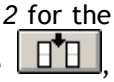


Fig. 18 Quick Tutorial. Adding a New Demand Pattern.

3. Enter the multiplier values *0.5, 1.3, 1.0, 1.2* for the time periods 1 to 4. Press Enter to add a new multiplier to the list. Use  , to insert or remove a multiplier respectively.

4. Click the **OK** button to accept the new pattern.

5. **Exit** the pattern editor.

The multipliers are used to modify the demand from its base level in each time period. Since we are making a run of 72 hours, the pattern will wrap around to the start once again after each 24-hour interval of time.

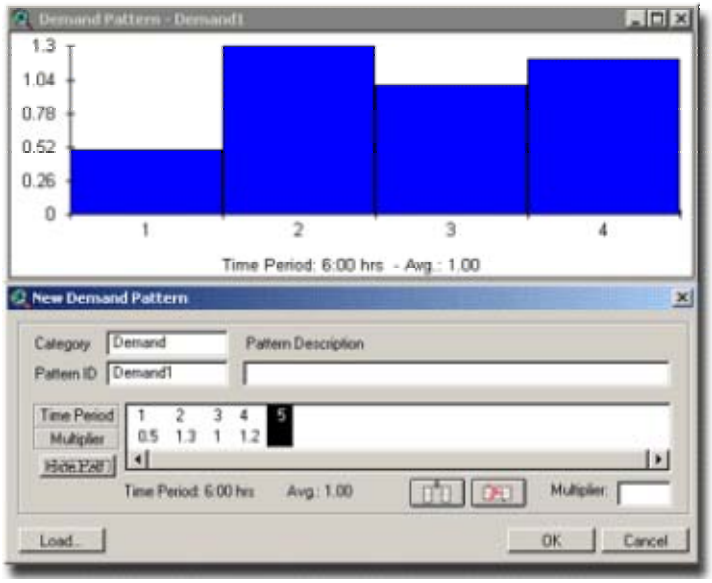


Fig. 19 Quick Tutorial. Demand Pattern Dialog. Viewing a Pattern Chart.

We now need to assign *Demand1* pattern to the **Demand Pattern** property of all of the junctions in our network. We can utilize one of GISRed's **Default Hydraulic Options** to avoid having to edit each junction individually. If you bring up **Scenario** | **Defaults** | **Hydraulics** in the menu bar you will see that there is an item called **Default Pattern**. Setting its value equal to ***Demand1*** will make the *Demand Pattern* at each junction equal *Demand1* providing no other pattern is assigned to the junction.

Alternatively, we can use the **Group Edit** dialog to assign the pattern to all Junctions.
In that case:

1. Select all junction with the '**Select Feature**' tool . (Make sure the node theme is the active theme).

2. Bring up **Edit Model** | **Group Edit…**

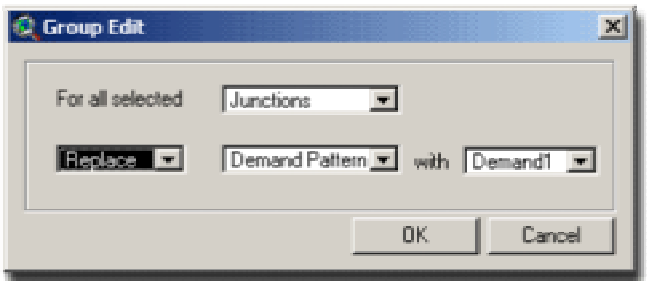


Fig. 20 Quick Tutorial. Group Editing.

3. Select the property '*Demand Pattern*' and *Demand1* from the dropdown list.

4. Accept.

GISRed shows a report window specifying the number of junctions affected by the change.

1.13 RUNNING AN EXTENDED PERIOD SIMULATION

We are now ready to run the extended period hydraulic analysis. Once again select **Scenario | Export Epanet Input File** (or click the button).
Once the File has been successfully created, select **Scenario | Run Analysis** and choose the file that is just been made.

For extended period analysis you can select the time at which you want to see the results. To do that, use the dropdown list or simply press the buttons of the slide control.

For example, to see pressure and flow results:

1. Choose **Pressure** in the '*Nodes'* dropdown list. Notice how a new temporal node theme appears on the TOC of the scenario.

2. Choose **Flow** in the '*Links'* dropdown list. Notice how a new temporal link theme appears on the TOC of the scenario.

3. Select a different time either from the dropdown list or from the slider. Notice how the result themes of the scenario update their values.

4. To make a temporary result theme permanent, use the '*Save Theme'* button. This will allow you to save a specific set of results to a permanent theme.

Note the periodic behavior of the water elevation in the tank over time.

1.14 OTHER THINGS

We have only touched the surface of GISRed's capabilities. Some additional features of the extension that you should experiment with are:

- Editing any property for a group of selected objects
- Using Control statements to base pump operation on time of day or tank water levels
- Exploring different Legend Options, such as changing the node size, the type of classification, etc
- Creating a catalog image and adding an aerial photograph to digitize the network over the image
- Using the overview map
- Interpolating elevations
- Allocating nodal demands
- Setting up a calibration configuration with the calibration manager
- Exporting the scenario to a image file
- Creating customized layouts
- Querying the network (current nodal demands, pipe roughness values, diameters, etc.)
- Checking the connectivity…

CHAPTER 2. GISRED'S GUI (GRAPHICAL USER INTERFACE)

This chapter discusses the essential features of GISRed's workspace. It describes all the menus and tools that the extension adds to the ArcView environment.

The GISRed Extension extends the ArcView interface by adding a series of new menus and dialogs to assist the user in the process of making a water distribution network model.

2.1 EXTENSION WORKSPACE

The basic GISRed workspace is pictured below. It consists of the following extended user interface elements: a new **Document**, four additional **Menus** and eight **Tools**. A description of each of these elements is provided in the sections that follow.

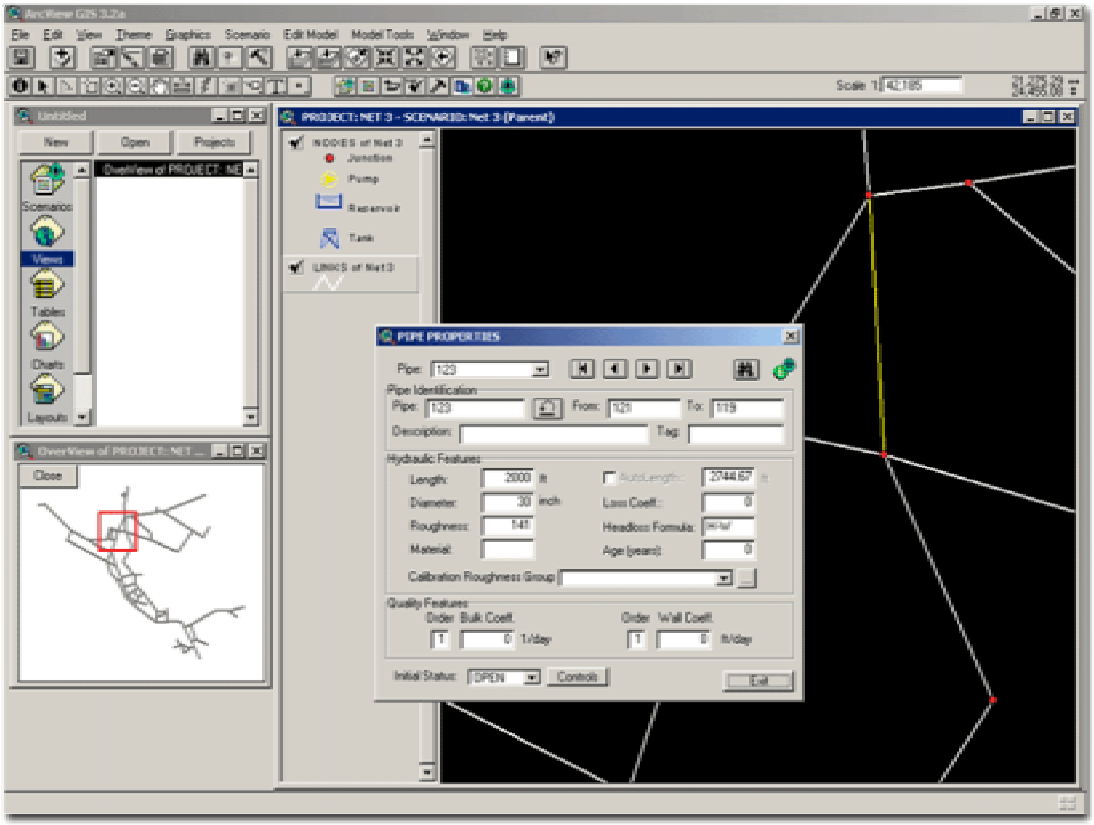


Fig. 21 GISRed Workspace.

2.1.1 PROJECT GUI (ArcView Project Manager)

When the extension is loaded, a new document is added to the ArcView Project Manager Window. This document is basically a clone of the View Document with some caveats. Let us see the main differences:

Scenarios Document

When the Scenarios Document is active, the ArcView Project Manager Window is customized with three buttons:

New

Creates a new GISRed project and its corresponding parent Scenario. Also double-clicking the Scenarios icon. This button is always active.

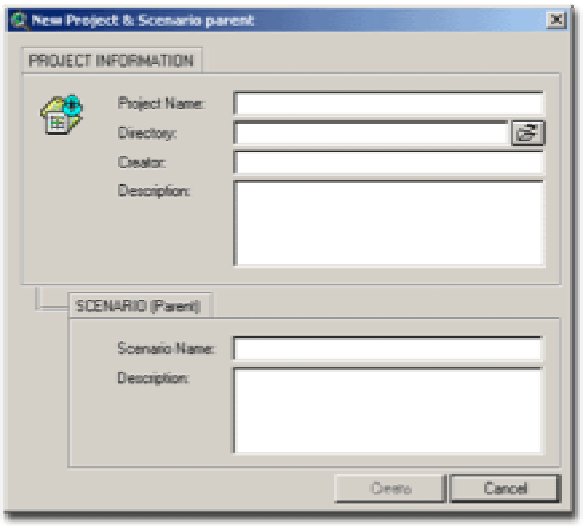


Fig. 22 New GISRed Project Dialog.

Open

Opens the scenario selected in the *ArcView Project Manager Window* and makes the project it belongs to the current project. This button is active just when a scenario is selected.

Projects

Opens the GISRed Project Manager. This button is always 'on'.

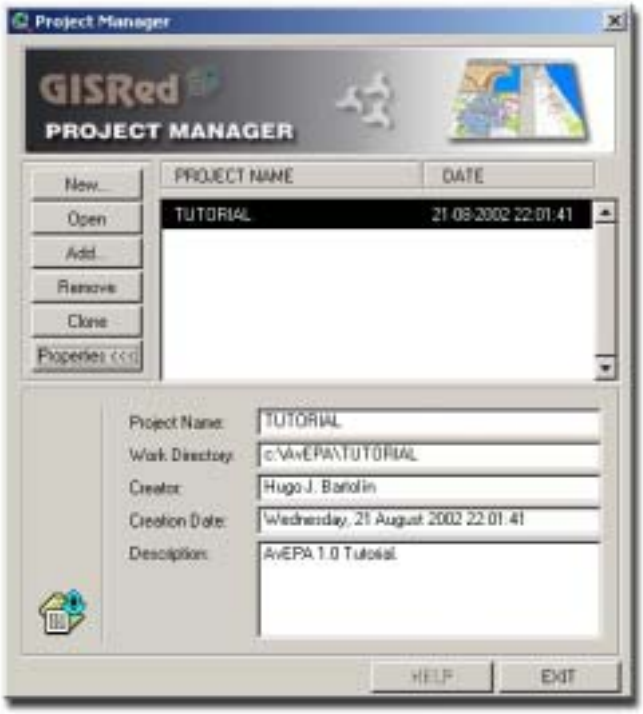


Fig. 23 GISRed Project Manager.

2.1.1.1 GISRed Menu

If the ArcView Project Manager Window is the active window, the menu bar shows a new menu called GISRed.

New
Creates a new GISRed project and its corresponding parent Scenario.

Import
Shows the Network Importation Dialog.

Project Manager
Opens the GISRed Project Manager.

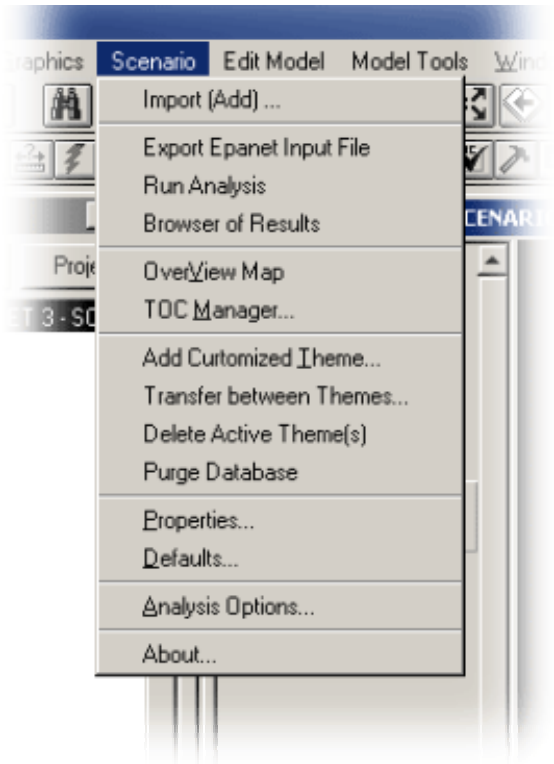
About…
Shows information about the version of the extension and the authors.

2.1.2 SCENARIOS GUI (GISRed environment)

Three new menus and eight new tools are added to the Scenario GUI.

2.1.2.1 MENUS

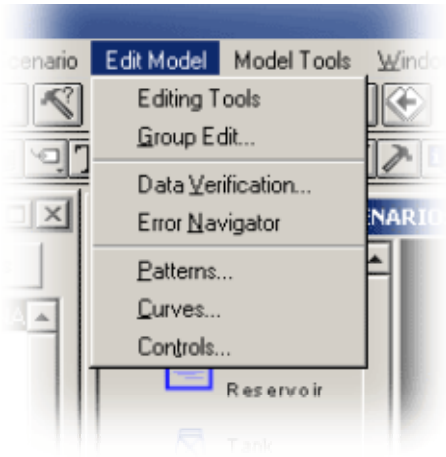
Scenario Menu



The Scenario Menu contains the options for managing scenario documents, adding new themes to the scenario, getting the scenario properties, importing data into the scenario, etc.

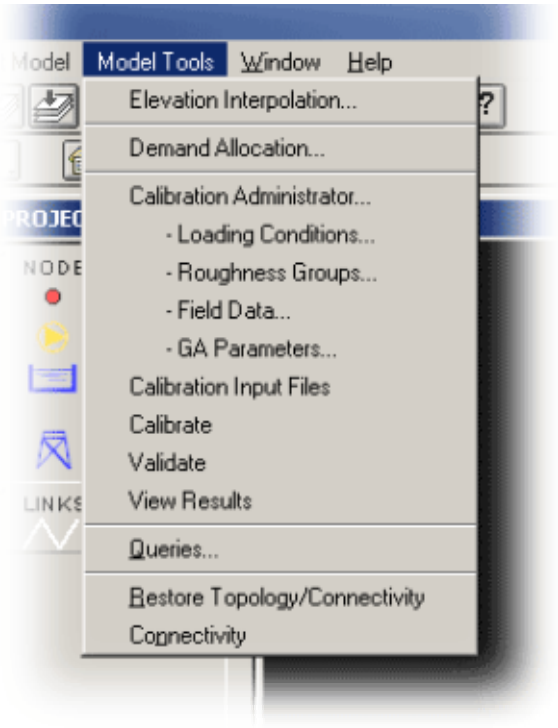
OPTION	DESCRIPTION
Import (Add)	Imports and adds a network data into the current scenario.
Export Epanet Input File	Exports an EPANET Input File.
Run Analysis	Performs an EPANET Simulation.
Browser of Results	Shows the Browser window
Overview Map	Toggles the Overview Map on/off.
TOC Manager	Opens the TOC Manager.
Add Customized Theme	Opens the GISRED Theme Manager
Transfer between Themes	Allows transference of information from a basic model-oriented network theme to a management-oriented network theme and viceversa.
Delete Active Theme(s)	Deletes a theme from the TOC.
Purge Database	Deletes all non-referenced themes from the project database.
Properties	Shows a summary of the current network model.
Defaults	Opens the default properties dialog
Analysis Options	Edits analysis options
About	Opens the About window which shows the Extension version and the authors.

Edit Model Menu



OPTION	DESCRIPTION
Editing Tools	Sets the Properties Edition Mode for the currently active theme.
Group Edit	Edits a property for the group of objects that are currently selected within the scenario.
Data Verification	Opens a dialog that allows to check out the connectivity of the network (spatially or topologically) and some network properties.
Error Navigator	Opens a navigator window to check out visually each one of the errors associated with the error theme currently active.
Patterns	Opens the Pattern Editor.
Curves	Opens the Curve Editor.
Controls	Opens the Controls Editor.

Model Tools Menu



OPTION	DESCRIPTION
Elevation Interpolation	Opens the dialog for the elevation interpolation process. Requires a Grid Theme on the TOC and consequently Spatial Analyst Extension.
Demand Allocation	Opens the main dialog to perform a demand allocation process going through a series of dialogs.
Calibration Administrator	Opens the Calibration Administrator dialog.
Loading Conditions	Allows to choose the instants of time for which the network is going to be calibrated. This option applies to the current network calibration configuration.
Roughness Groups	Allows the definition of the calibration groups in terms of roughness. This option applies to the current network calibration configuration.
Field Data	Allows to import field data for the actual calibration configuration. This option applies to the current network calibration configuration.
GA Parameters	Allows to define all the genetic algorithm parameters. This option applies to the current network calibration configuration.
Calibration Input Files	Generates all the input files required by the calibrator to run a calibration. This option applies to the current network calibration configuration and the active scenario.
Calibrate	Launches the calibrator. This option needs the calibrator input files.
Validate	Launches the validation process after a calibration has been performed.

View Results	Opens the calibration-related results dialog.
Queries	Allows to perform network queries generating legend classified themes such as diameter, roughness, age,…
Restore Topology/Connectivity	Restores the topology and connectivity of the network model.
Connectivity Analysis	Performs a connectivity analysis on the network. Allows to check out disconnected areas

2.1.2.2 BUTTONS AND TOOLS

These buttons/tools provide shortcuts to commonly used operations.

ABOUT: Shows information about the version of the extension and the authors.

IMPORT: Opens the network importation dialog.

ADD GISRED THEME: Opens the GISRed theme manager.

TOC MANAGER: Opens the TOC Manager Dialog.

EDITING TOOLS: Shows the editing tool bar associated with the current theme.

PROPERTIES: Sets the *Properties Edition Mode* for the currently active theme.

NETWORK QUERIES: Opens the network query dialog.

EXPORT TO EPANET: Generates an EPANET input file. NOTE:

2.1.2.3 NETWORK SCENARIO

The main concept of the GISRed Extension is the scenario document. The scenario is the object that will show all the graphical information stored in a relational database, by means of two basic themes, namely the node and link feature themes, which are the foundations of what is expected to be the water network model.

The Network Scenario provides a planar schematic diagram of the objects comprising a water distribution network. The location of objects and the distances between them do not necessarily have to conform to their actual physical scale, even though it is customary to work with a specific projection and map units defined in the **View | Properties** menu. The aforementioned basic network themes can be displayed by using different symbols, colors, sizes, etc. The color-coding is described in the associated Legend, which can be edited. New objects can be directly added to the map and existing objects can be clicked on for editing, deleting, and repositioning. New themes (such as a street or topographic map) can be added to the Scenario TOC (table of contents) for reference. The scenario can be customized, zoomed to any scale and panned from one position to another as would be done for a view document using the ArcView tools. The scenario can be used to create customized layouts or exported as a DXF file, Windows metafile or image.

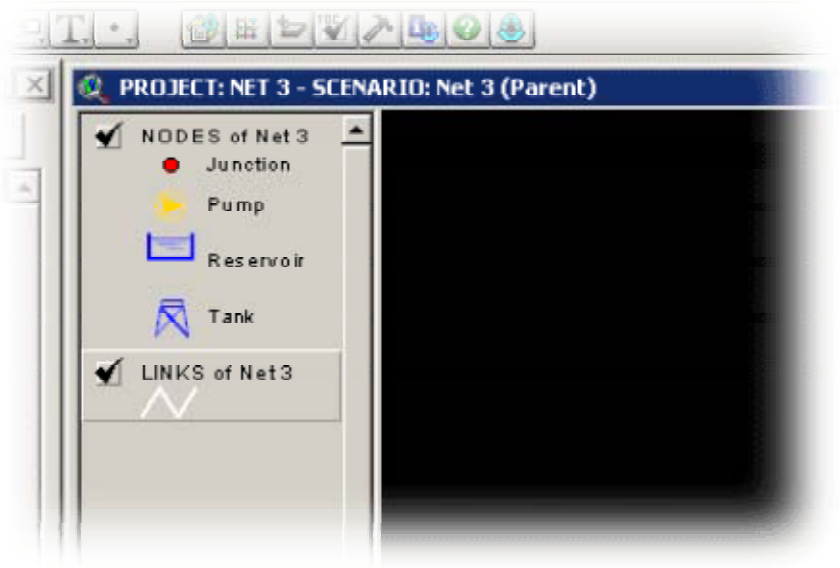


Fig. 24 Scenario.

CHAPTER 3. WORKING WITH GISRED PROJECTS

This chapter discusses how GISRED uses a project database to store all network model data. Moreover, it explains how to work with the GISRED Project Manager to handle new/existing projects and set certain default options for them.

3.1 GISRED PROJECT MANAGER

There are two ways to open the GISRED Project Manager. Firstly, make the ArcView Project window the active window and then use:
either

1. **GISRed | Project Manager**

or

2. **Projects** button that appears when the ***Scenarios*** document is the active document in the ArcView Project window.

Both ways open the following dialog:

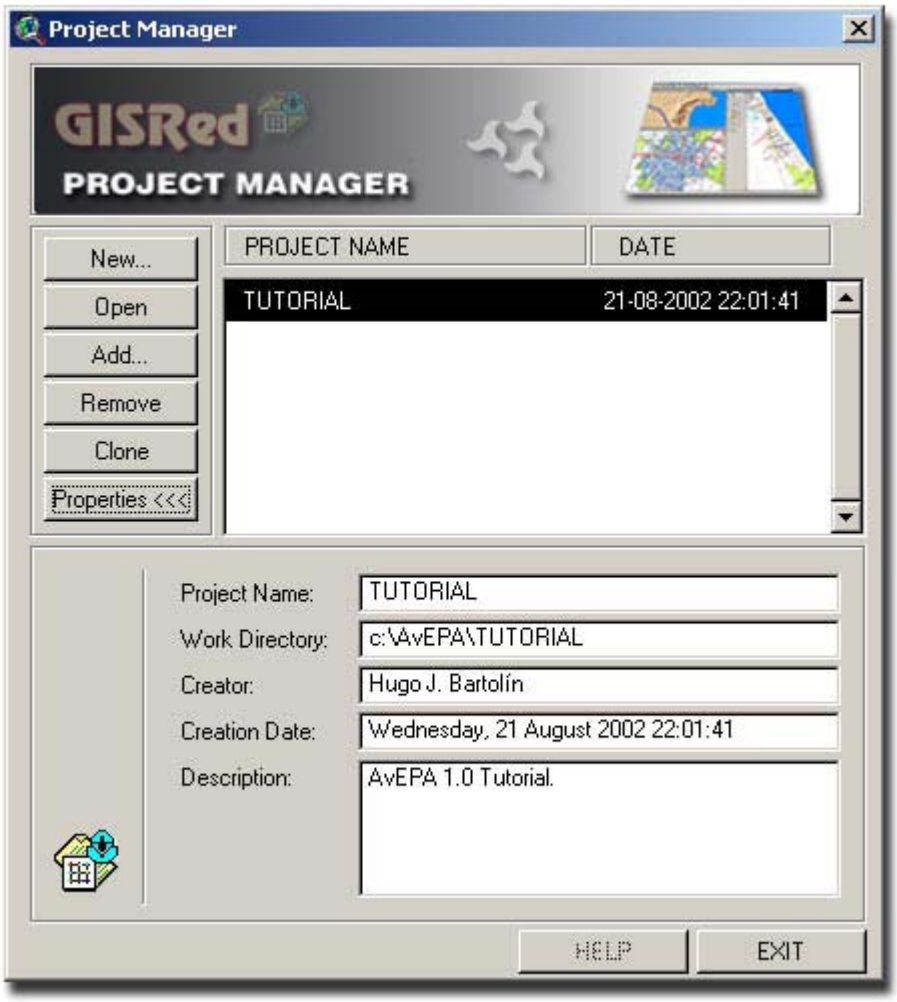


Fig. 25 GISRed Project Manager Dialog.

This dialog presents all projects created with the extension. To see more details about the project, select one from the list and press ***Properties >>>***, the dialog is expanded and information about the project is shown. It is possible to edit some of the properties; if any editable property is modified, you must accept the changes pressing the **Accept** button that emerges.

The rest of the buttons are now explained:

NEW
Creates a new GISRed Project and add it to the list of projects. When the user clicks on this button a new dialog comes up requiring all the information for the new project and its parent scenario.

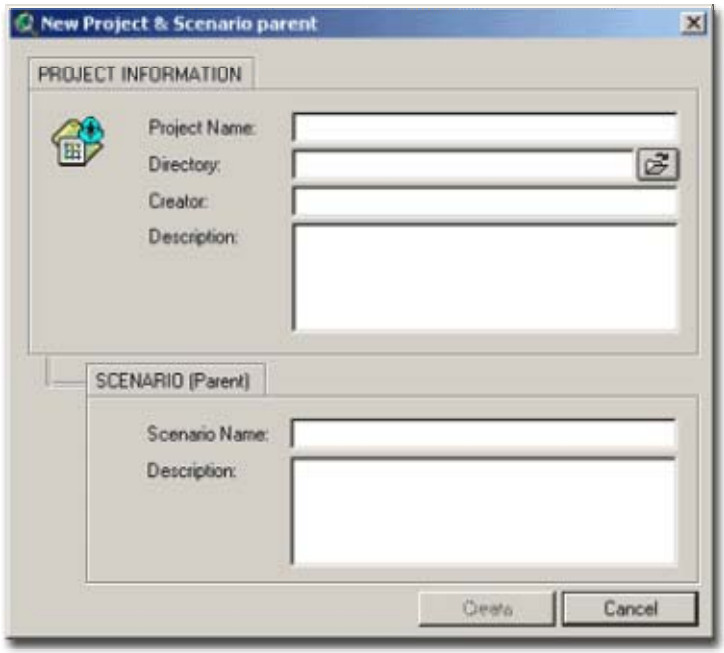


Fig. 26 Creating a New GISRed Project.

When a new project is created, a relational database is generated along with it. This database is placed in a folder named as the ***Project Name*** in the path specified in the ***Directory*** textline. At first all the tables of the database are empty. The basic tables of a new project are:

Shapefiles:
links1.shp, links1.shx, links1.dbf, nodes1.shp, nodes1.shx, nodes1.dbf
Dbase Tables:
Analysis.dbf, Curves.dbf, default.dbf, Ids1.dbf, JUNCT1.dbf, Master.dbf, Mpipes1.dbf, MULTDM.dbf, OBJDSC.dbf, OBJTYP.dbf, PATT.dbf, PUMPS1.dbf, RESERV1.dbf, ScnID.dbf, SMPCNTR.dbf, SRCQUL.dbf, Tanks1.dbf, Valves1.dbf

OPEN
Opens the parent scenario of the selected project.

ADD
This option allows to import a project

REMOVE
Removes the selected project from the list. It does not remove the project from the disk.

CLONE
Makes a clone of the selected project. The new project is automatically named as the original with the suffix '*(Clone)*'. The user can always change the name later on.
Note: When cloning a project, a copy of the original database is created (project folder and all the tables and shapefiles that contains).

PROPERTIES
Expands or retracts the project manager window.

3.2 OPENING AND SAVING PROJECTS

An GISRED Project contains all the information used to model a network. A project typically consists of a folder (named as the user specified when created) with a series of tables and shapefiles which contain all the details of the network.

To create a new GISRed project:

1. Open de **GISRed Project Manager**. Select **New**. Alternatively, double click the **Scenarios Icon** of the *ArcView Project Window*.
2. You will be prompted to enter all the information related to the new project before it is created.
3. A new project is created with all options set to their default values.

NOTE: A new unnamed project is automatically created whenever ArcView GIS first begins. The ArcView Project comprises all documents in a session, namely, all views, layouts, tables, etc. of the current session. An ArcView project is saved to a structured text file with the extension 'apr'.
An GISRed project does not need to be saved, since any change in the database is automatically committed. However, if the user wants to keep the whole working session 'as is' (scenarios, views, layouts, scripts, tables, customizations, etc.), to be used later on, it is necessary to save the whole project in a ArcView **.apr* file.

To save an ArcView session (GISRed scenarios included) on disk:

1. **Select File | Save Project As…**
2. Specify a path and a name for the project.
3. The Project will be stored on disk with the extension **.apr* as a conventional ArcView project.

To open an existing **.apr* project stored on disk:

1. Open ArcView.
2. Either select **File | Open Project…** from the Menu Bar or click on the Standard Toolbar.
3. Select the file to open from the Open File dialog form that will appear.
4. Click OK to close the dialog and open the selected file.

3.3 PROJECT DEFAULTS

Each GISRed project has a set of default values that are used unless overridden by the user. These values fall into three categories:

- **Default ID labels** (labels used to identify nodes and links when they are first created)
- **Default node/link properties** (e.g., node elevation, pipe length, diameter, and roughness)
- **Default hydraulic analysis options** (e.g., system of units, headloss equation, etc.)

To set default values for a project:

1. Select **Scenario | Defaults** from the Menu Bar.
2. A Defaults dialog form will appear with three radio buttons, one for each category listed above.

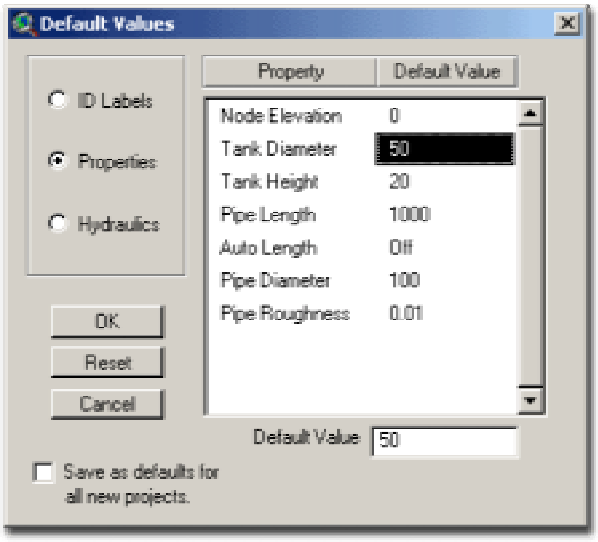


Fig. 27 GISRed Projects Defaults.

3. Choose a category and click on the ***Default Value*** *column* of the *ListBox* to edit the value.
4. Change the value in the ***Default Value*** *Textline* and commit the value pressing *Enter*. The value should then change in the listbox.
5. Check the box in the lower left of the dialog form if you want to save your choices for use in all new future projects as well.
6. Press **Reset** to change all defaults to original values.
7. Click **OK** to accept your choice of defaults.

The specific items for each category of defaults will be discussed next.

DEFAULT ID LABELS

The ID Labels page of the Defaults dialog form is used to determine how GISRed will assign default ID labels to network components when they are first created. For each type of object one can enter a label prefix or leave the field blank if the default ID will simply be a number. As an example, if J were used as a prefix for Junctions, then as junctions are created they receive default labels of J1, J2, J3 and so on. After an object has been created, the Property Editor can be used to modify its ID label if need be.

DEFAULT NODE/LINK PROPERTIES

The Properties section of the Defaults dialog form sets default property values for newly created nodes and links. These properties include:

- Elevation for nodes
- Diameter for tanks
- Maximum water level for tanks
- Length for pipes
- Auto-Length (automatic calculation of length) for pipes
- Diameter for pipes
- Roughness for pipes

When the Auto-Length property is turned on, pipe lengths will automatically be computed as pipes are added or repositioned on the network map. A node or link created with these default properties can always be modified later on using the Property Editor.

DEFAULT HYDRAULIC OPTIONS

The third section of the Defaults dialog form is used to assign default hydraulic analysis options. It contains the same set of hydraulic options as the project's Hydraulic Options accessed through **Scenarios | Analysis Options | Hydraulics**. They are repeated on the Project Defaults dialog so that they can be saved for use with future projects as well as with the current one. The most important Hydraulic Options to check when setting up a new project are *Flow Units*, *Headloss Formula*, and *Default Pattern*. The choice of Flow Units determines whether all other network quantities are expressed in Customary US units or in SI metric units. The choice of Headloss Formula defines the type of the roughness coefficient to be supplied for each pipe in the network. The Default Pattern automatically becomes the time pattern used to vary demands in an extended period simulation for all junctions not assigned any pattern.

3.4 SCENARIO PROPERTIES

To view a summary description of the current scenario, select **Scenario | Properties** from the Menu Bar. The information dialog form will appear in which you can edit the title and description for the scenario as well as for the project it belongs to. The form also enables to display certain network statistics, such as the number of junctions, pipes, pumps, etc. To do that, just click on the '***Model Details***' button.

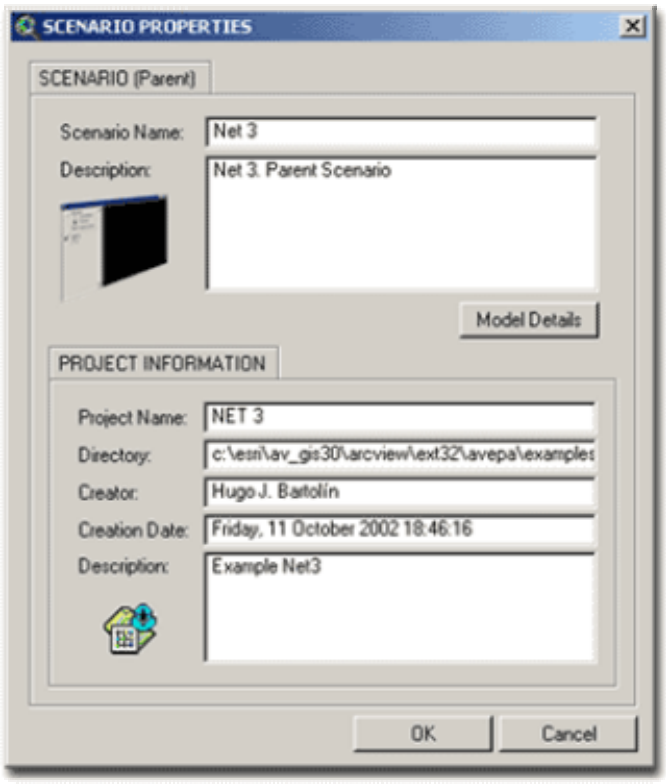


Fig. 28 Scenario Properties.

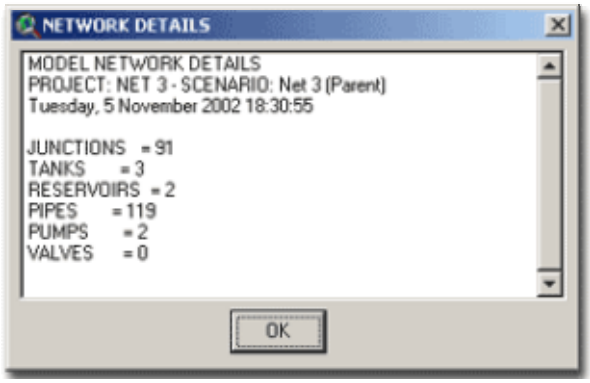


Fig. 29 Scenario Properties. Network Details.

CHAPTER 4. NETWORK ELEMENTS

GISRed uses various types of objects to model a distribution system. These objects can be accessed directly on the network scenario. This chapter describes what these objects are and how they can be entered, selected, edited, deleted, and repositioned.

4.1 TYPE OF NETWORK ELEMENTS

GISRed makes the difference between network elements in two basic themes. These elements, as part of a theme, appear on the network scenario. According to the theme they belong to, these elements can be classified as followed:

Node Theme:
- Junctions
- Reservoirs
- Tanks
- Pumps
- Valves

Link Theme:
- Pipes

4.2 EDITING TOOLS

There are a series of editing tools grouped in different toolbars that are directly dependant upon the current active theme. That is similar to the way of working within standard ArcView, in which the graphical user interface and menu options associated change depending on the document and current active theme of the project. This confers certain flexibility as well as an intuitive friendly environment. The toolbars that may be used with the application are:

LINK TOOLBAR	
DESCRIPTION	
The Link toolbar allows to introduce pipes to the model, edit and modify vertices, draw pipes by coordinates, split pipes at selected nodes and delete pipes. All these operations, preserving the topology and connectivity of the network.	

Tool	Function
	Adds a pipe (and the end junctions) using the mouse.
	Adds a pipe by entering the coordinates of all its vertices.
	Edits the pipe vertices.
	Deletes the selected pipes.

NODE TOOLBAR

DESCRIPTION

The Node toolbar allows to add junctions, pumps, valves, reservoirs and tanks, move nodes (and consequently all those pipes connected to that node), connect pipes to a node and delete nodes (merging the corresponding two pipes into just one). It will always be possible to convert any node from one to another type.

Tool	Function
	Clicking on a pipe it inserts a Junction splitting the pipe. Clicking on another node (pump, tank, valve,…), converts the node into a junction.
	Adds a Reservoir. Click on a junction to add a reservoir.
	Adds a Tank. Click on a junction to add a tank.
	Adds a Valve. Click on a pipe or junction to add a valve.
	Adds a Pump. Click on a pipe or junction to add a pump.
	Moves a junction. Whenever a junction is moved all pipes connected to it are moved as well.
	Connects the end of a pipe to a junction.
	Makes a ‘T’ connection. Connects the end junction of a pipe to a pipe (at any middle point) making a ‘T’ connection.
	Deletes the selected nodes (one or more) following the next rules: - Before any selected junction is deleted, the application checks whether the pipes connected to the junction can be merged or not. If they can, the junction is deleted. - When trying to delete a node other than junction, the node will be automatically converted to a junction and then the pipes merged if possible. **NOTES**: ➢ *Two pipes can only be merged if they have the same diameter, material and roughness.* ➢ *Obviously, junctions with more than two pipes connected to them cannot be deleted. In this case it is necessary to start deleting pipes.* ➢ *After a deleting operation the user is prompted with a summary window saying how many nodes could be deleted and how many could not.*

AUXILIARY PIPE TOOLBAR

DESCRIPTION	
The Auxiliary Pipe Toolbar is a toolbar for pipe themes not oriented to modelling. It works in a similar way as the Pipe Toolbar, but it does not take into account any topology action (pipes will not have end nodes). This toolbar is associated to a customized polyline theme controlled by the application. (*See section 10.3 to know how to add GISRed themes to the scenario*).	

Tool	Function
	Adds a pipe using the mouse.
	Adds a pipe by entering the coordinates of all its vertices.
	Edits the pipe vertices.
	Deletes the selected pipes.

IMAGE TOOLBAR

DESCRIPTION	
The Image Toolbar works with the image catalogs.	

Tool	Function
	Adds an image to an image catalog. See section 4.8.
	Deletes an image from a catalog by clicking on the corresponding image.

4.3 ADDING NETWORK ELEMENTS

4.3.1 Adding a Link

➢ To add a straight or curved pipe:

1. Make the link theme the active theme on the scenario.
2. Select **Edit Model | Editing Tools** or alternatively click the button to open the Link Toolbar if it is not already opened.
3. Click the button to set the adding pipe mode.
4. On the map, click the mouse over the pipe's start node.
5. Move the mouse in the direction of the pipe's end node, clicking it at those intermediate points where it is necessary to change the pipe's direction (vertices).
6. Double click the mouse over the pipe's end node.

NOTES:

- *The snap mode is automatically set to 'interactive' when digitizing pipes. The snap tolerance is 1/100 the scenario display width.*
- *Every time a pipe is added, GISRed checks whether the starting and end point of the pipe lie on existing nodes within a fixed distance tolerance equivalent to 3 screen pixels, if any of them (or both) does not, GISRed does automatically insert the corresponding end junction. Otherwise the end of the pipe gets connected to the existing node.*
- *Pressing the right mouse button while drawing a pipe will open a popup menu with additional operations such as zooming or panning the scenario.*
- *If the pipe is not connected to the rest of the network, the application prompts the user with a warning message ('A new network might be originated').*

4.3.2 Adding a Node

To add a Node, use the Node Toolbar:

1. Make the node theme the active theme on the scenario.
2. Select **Edit Model | Editing Tools** or alternatively click the button to open the Node Toolbar if it is not already opened. There is also a third possibility which is the *popup menu*, to do this, right click and choose **Editing Tools**.
3. Choose the element you want to add (junction, reservoir, tank, valve or pump)

➢ *For Junctions, Valves and Pumps*:

4. Move the mouse to the desired location on a pipe and click.
5. The element is inserted at that point.

➢ *For Tanks and Reservoirs*:

6. Move the mouse to the desired junction and click the mouse.
7. The tank/reservoir is inserted at that point.

NOTES:

- *Junctions can only be added on pipes.*
- *Tanks and Reservoirs can only be added at existing junctions. Prior to entering any tank or reservoir a junction must exist.*
- *Pumps and valves can be added either at a junction (connectivity degree lower than 2) or in the middle of a pipe.*

4.4 SELECTING ELEMENTS

To select (a) network element(s) on the scenario:

1. Make sure that the scenario is the active window.
2. Make the corresponding theme active.

➢ Using the **'select feature'** tool :

a. Click the tool.
b. Click/Drag the mouse over the desired element(s) on the scenario.
c. Use the ***shift key*** to add elements to the selection.

➢ Using the ArcView '***query builder***' :

The *Query Builder* lets you select features by defining a query based on their attributes.

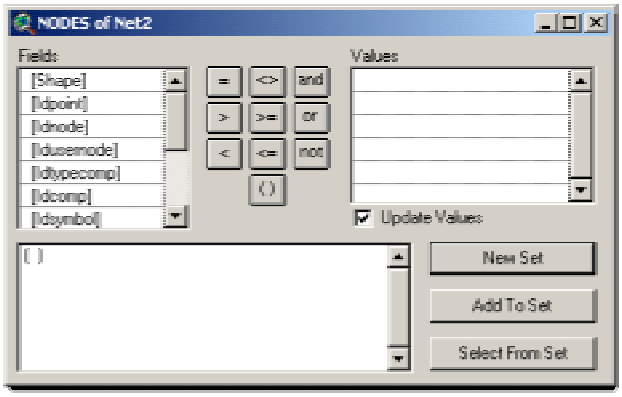


Fig. 30 ArcView Query Builder Dialog.

To build a query, choose a Field, then an Operator, then a Value. You build a query by double-clicking on these options with the mouse or by typing your query directly into the query text box. By default, the query is contained within parentheses, but the parentheses may not be required, depending on the complexity of your query. If the Update Values choice is on, click once on a field name to list its values in the Values list. Field names are always enclosed in square brackets ([ ]). If the value you want to use in the query is not in the Values list, type it into the query text box.

- **New Set**: Makes a new selected set containing the features or records selected in your query. Features or records not in this set are deselected.
- **Add To Set**: Adds the features or records selected in your query to the existing selected set. If there is no existing selected set, the features or records specified in the query become a new set. Use this option to widen your selection.
- **Select From Set**: Selects the features or records in your query from the existing selected set. Only those features or records in this existing set that are selected in your query will remain in the selected set. Use this option to narrow down your selection.

➢ Using the '***Select by Theme…'*** menu:

Selects features in the active theme(s) based on their location in relation to the selected features in another theme. You can also use this dialog to select features in the active theme based on their location in relation to the selected features in the same theme.

To select a theme using another theme:

1. Make the theme active that contains the features you wish to select.
2. In the ***Theme*** menu, choose ***Select By Theme***.
3. Choose the spatial relation type.
   ArcView provides several spatial relation types. The types you see depend on the selector theme's feature type. In some cases, you may want to pick the selector theme before choosing the spatial relation type.
4. Pick the theme whose features to use to make the selection.
   This is the selector theme. ArcView selects features in the target themes using the selected features of the selector theme. If no features in the selector theme are selected, ArcView uses all the selector theme's features to select.
5. If you chose *Are Within Distance of*, specify the selection distance.
   ArcView selects the features in the target themes whose distance to any of the features in the selector theme is less than or equal to the selection distance. The selection distance units are the distance units of the scenario. These units are set by choosing Properties from the View menu.
6. Choose a selection method.
   ArcView provides three selection methods. *New Set* creates a new set of selected features from all candidate features. *Add to Set* adds new features to the existing selected set of features. *Select from Set* selects features only from candidates in the currently selected set of features.

NOTES:

- Specifying a selection distance essentially creates a buffer polygon around the features of the selector theme; however, the buffer polygon is not visible.
- If you specify a selection distance of 0, ArcView selects the features that intersect the selector theme's features. A selection distance of 0 selects all points in the target themes that fall within a polygon, all lines that intersect, and all lines that intersect the selector theme's polygons and all polygons that intersect the selector theme's lines.
- ArcView uses the selected features of a theme to make the selection; therefore, if the active theme is the same theme as the selector theme, and you choose Intersect, you can select features that are adjacent to the selected features.

➢ Using the '***Select Features using Graphic'*** button :

The ***Select Features Using Graphic*** button lets you select features that fall inside or are intersected by shapes such as circles, boxes, lines and polygons that you have already drawn on a view using the ***Draw tools***. Features will be selected from all of the currently active themes.

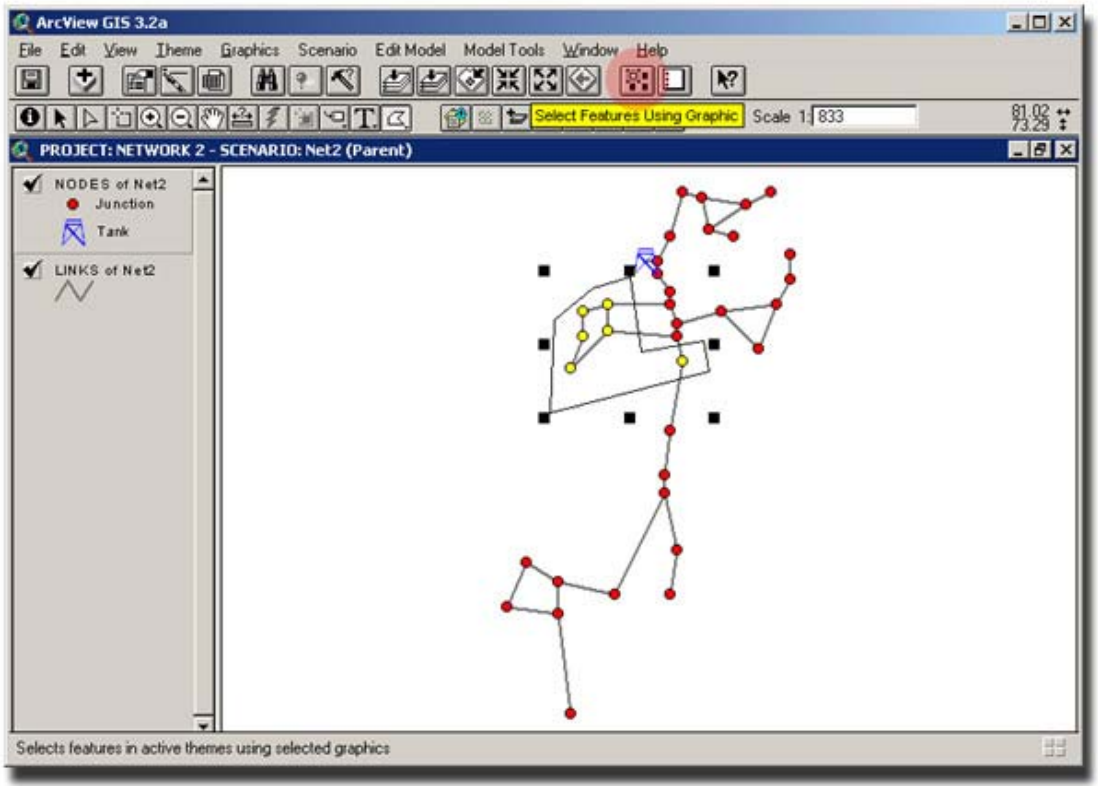

ArcView GIS 3.2a
File Edit View Theme Graphics Scenario Edit Model Model Tools Window Help
Select Features Using Graphic
Scale 1: 833
PROJECT: NETWORK 2 - SCENARIO: Net2 (Parent)
NODES of Net2
Junction
Tank
LINKS of Net2
Selects features in active themes using selected graphics

Fig. 31 Selecting features using graphics.

➢ To Select features by a graphic:

a. Draw the graphic (line, polygon,…).
b. Keep the graphic selected.
c. Click the button .
d. Click/Drag the mouse over the desired element(s) on the scenario.
e. Use the ***shift key*** to add new elements to the selection by using additional graphics.

➢ Using the '***Find***' button :

Find is a quick way to select a particular feature of interest. For example, if you are working with a theme of links and you want to select a pipe with a specific name, simply type that name into the Find dialog box. Find only lets you select one feature at a time.

4.5 DELETING AN ELEMENT

➢ To delete Pipes:

1. Select the pipe(s) on the scenario using one of the aforementioned options.
2. Open the Link Toolbar.
3. Press the Delete button .
4. The deletion will be confirmed before it takes effect.

➢ To delete a node:

1. Select the node(s) on the scenario using one of the aforementioned options.
2. Open the Node Toolbar.
3. Press the Delete button .
4. The deletion will be confirmed before it takes effect.
5. Once the deletion is confirmed, the application shows a summary of the elements that were deleted.

Selected nodes are deleted following the next rules:

- Before any selected junction is deleted, the application checks whether the pipes connected to the junction can be merged or not. If they can, the junction is deleted.
- When trying to delete a node other than junction, the node will be automatically converted to a junction and then the pipes merged if possible.

NOTES:

- *Two pipes can only be merged if they have the same diameter, material and roughness.*
- *Obviously, junctions with more than two pipes connected to them cannot be deleted. In this case it is necessary to start deleting pipes.*
- *After a deleting operation the user is prompted with a summary window saying how many nodes could be deleted and how many could not.*

4.6 MOVING A JUNCTION

GISRed only allows to move junctions.

➢ To move a node to another location on the scenario:

1. Open the Node toolbar.
2. Select the **Move Junction** tool .
3. Click the junction to move.
4. Click the new location for the junction.
5. All links connected to that junction are moved as well.

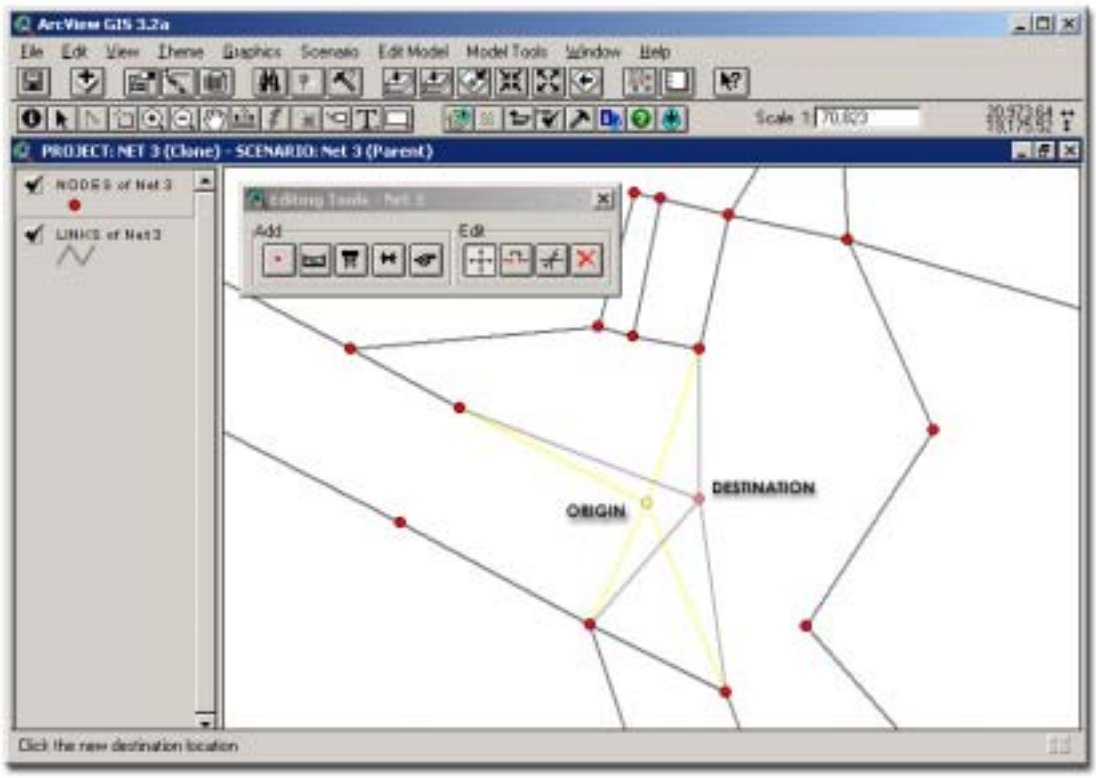

ArcView GIS 3.2a
File Edit View Theme Graphics Scenario Edit Model Model Tools Window Help
PROJECT: NET 3 (Clone) - SCENARIO: Net 3 (Parent)
NODES of Net 3
LINKS of Net 3
Add
Edit
DESTINATION
ORIGIN
Click the new destination location

Fig. 32 Moving a Node.

4.7 OTHER OPERATIONS

4.7.1 Modifying Pipe Layout

To reshape any of the pipes, use the tool of the link toolbar:

1. First click on the pipe to select it. Automatically a new red dashed outline shows up.
2. New vertices can be added to the outline just by clicking on it.
3. To move a vertex, place the pointer on it (the cursor will change) and then drag it with the left mouse button held down to its new position.
4. To delete a vertex, place the pointer on it and use the '***Del'*** key.
5. Click anywhere on the scenario to accept the new shape.

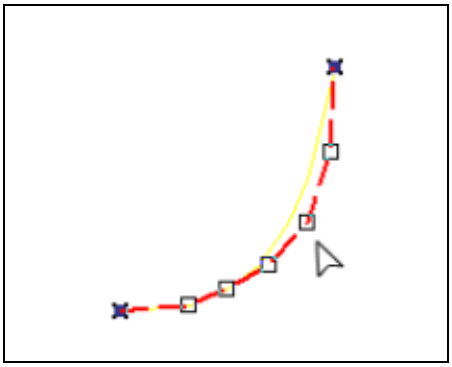


Fig. 33 Modifying Pipe Layout.

4.7.2 Connecting Pipes

➢ To connect an end junction of a pipe (only those with connectivity degree of 1) to another junction:

1. Make the node theme the active one.
2. Open the node toolbar.
3. Click on the tool .
4. Click on the junction you want to connect (origin). The pipe will be highlighted.
5. Click on the destination junction.
6. The pipe will be connected to the destination junction.

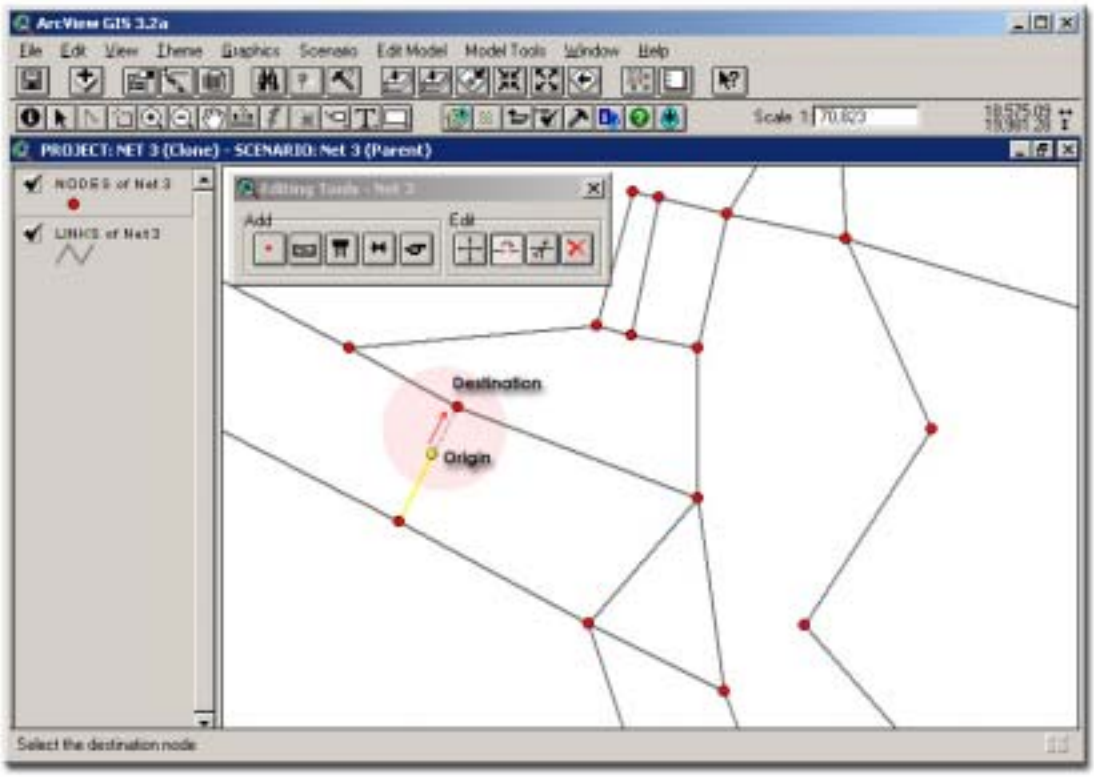

ArcView GIS 3.2a
File Edit View Theme Graphics Scenario Edit Model Model Tools Window Help
PROJECT: NET 3 (Clone) - SCENARIO: Net 3 (Parent)
NODES of Net 3
LINKS of Net 3
Add
Edit
Destination
Origin
Select the destination node

Fig. 34 Connecting Pipes.

4.7.3 Creating a T Connection

➢ To connect an end junction of a pipe (only those with connectivity degree of 1) to any point of another pipe by creating a '**T**' connection:

1. Make the node theme the active one.
2. Open the node toolbar.
3. Click on the tool .
4. Click on the junction you want to connect (origin). The pipe will be highlighted.
5. Click on the destination point of the pipe (it cannot be a node).
6. A '**T**' connection is automatically created.

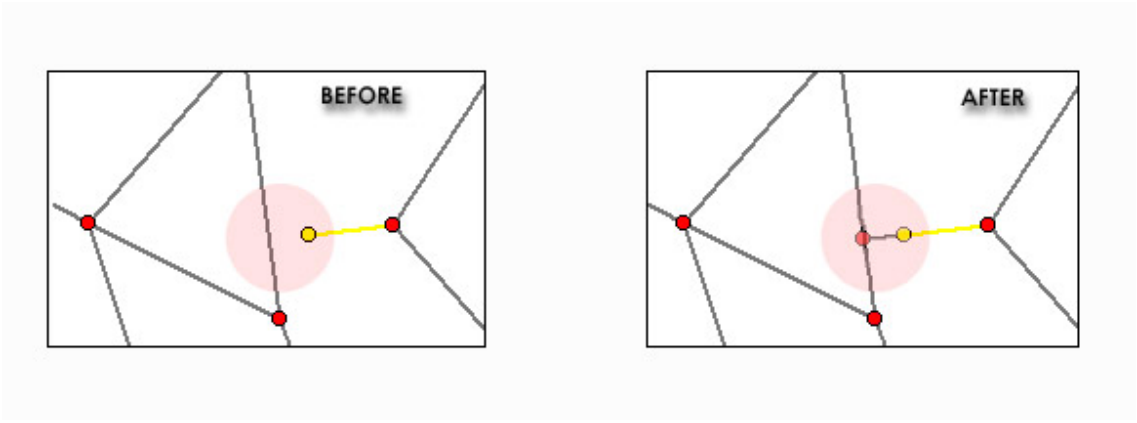


Fig. 35 Creating a T connection.

4.8 CREATING AN IMAGE CATALOG IN GISRED

GISRed allows to add images to the scenario by means of an *image catalog theme*. An image catalog is an organized collection of spatially referenced geographic images that can be accessed as one logical image theme. Image catalogs typically contain images that depict the same thematic information for a given geographic area of interest.
It is possible to add as many catalog themes as required. The image catalog is stored in a dBASE file.
The legend of image theme created from an image catalog is based on the first image found in the image catalog. All legend settings from the first image in the image catalog are used for each image in the image catalog.

➢ To add a catalog theme:

1. Use the menu option '**Add Customized theme…**' or click on .
2. Choose ***Image Catalog*** in the *Category combo box*.
3. Choose an ***Image Format***.
4. Enter a *Name* for the Catalog.

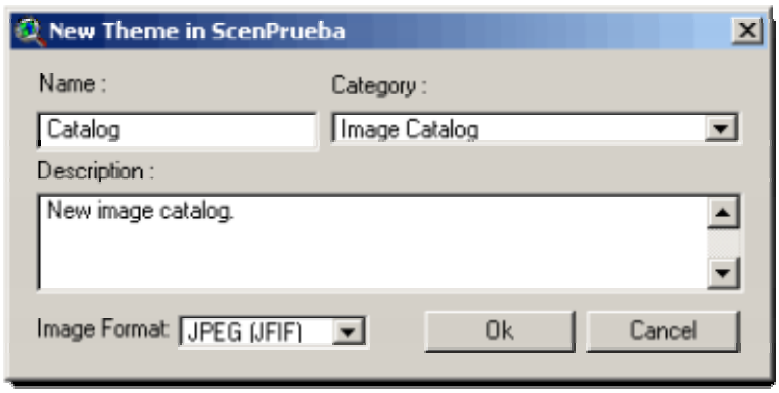


Fig. 36 Adding an image catalog to the scenario.

5. Press OK.

The application asks for the first image to be added to the Catalog. Once the user chooses one image file, GISRed opens the image in a new view and shows the ***Tic Selection Dialog***.

Now, the user has to enter the matching points between the image of the view and the scenario. To do that:

7. Click the **first** point on the **view**. The '*Tic Selection dialog*' shows the coordinates of the selected point. It is possible to type the coordinates directly on the dialog.
8. Click the **first** point on the **scenario**. *Idem*.
9. Click the **second** point on the **view**. *Idem*.
10. Click the **second** point on the **scenario**. *Idem*.

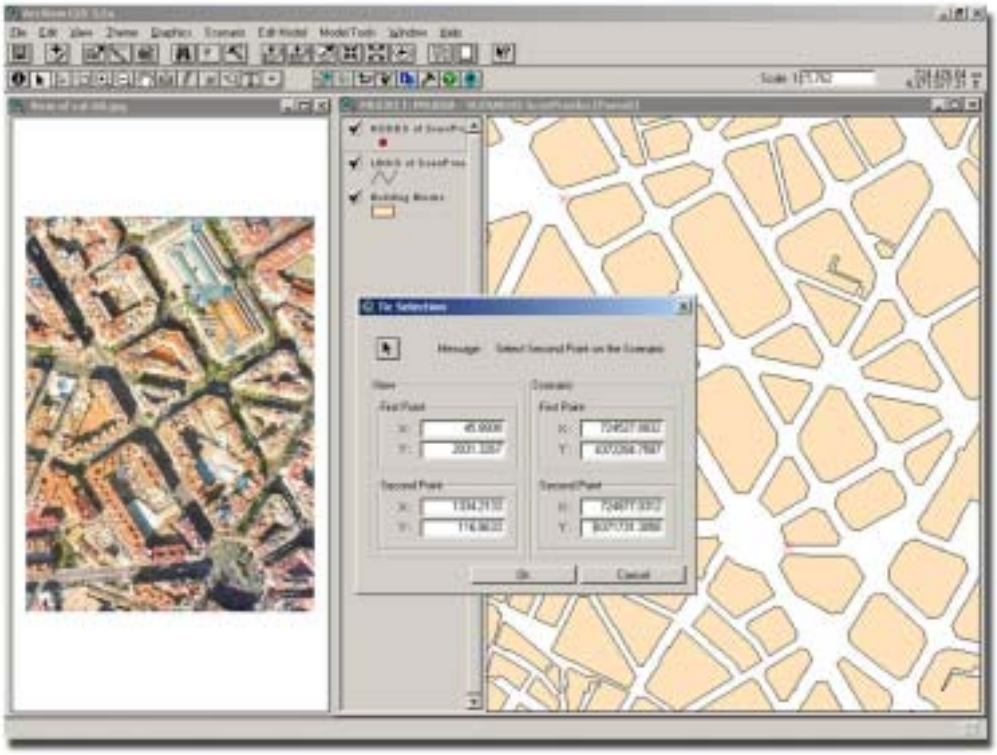


Fig. 37 First image to the Catalog. Tic Selection Dialog.

11. Press **OK** to create the image catalog and add it to the scenario.

Now, it is possible to use the image as a reference to add pipes and other elements of the network.

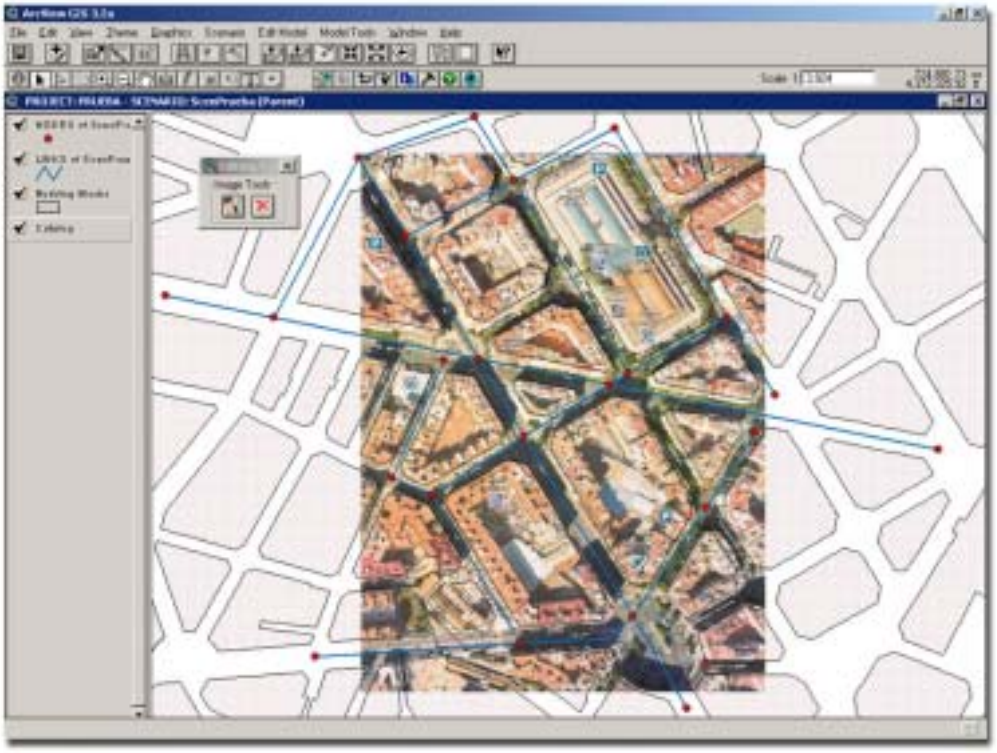


Fig. 38 Image Catalog Theme.

4.8.1 ADDING AN IMAGE TO AN IMAGE CATALOG

Once the image catalog is created, it is possible to add more images to it.

➢ To add a new image to the image catalog:

1. Make the image catalog active in the TOC.
2. Open the image toolbar. To do this, select **Edit Model | Editing Tools** or click the button or use the popup menu.
3. Press .

   The application asks for a new image to be added to the Catalog. Once the user chooses the image file, the image is opened in a new view and the ***Tic Selection Dialog*** shows up.

Now, the user has to enter the matching points between the new image of the view and the scenario. To do that:

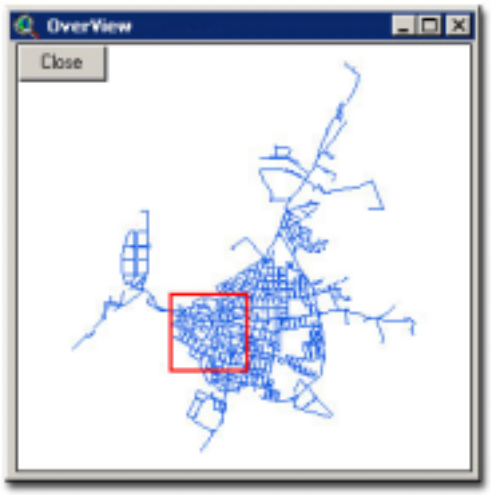
4. Click the **first** point on the **view**. The '*Tic Selection dialog*' shows the coordinates of the selected point. It is possible to type the coordinates directly on the dialog.
5. Click the **first** point on the **scenario**. *Idem*.
6. Click the **second** point on the **view**. *Idem*.
7. Click the **second** point on the **scenario**. *Idem*.
8. Press **OK** to create the image catalog and add it to the scenario or **Cancel** to close the image view and cancel the process.
9. If OK was pressed, the image is added to the catalog.

4.8.2 DELETING AN IMAGE FROM AN IMAGE CATALOG

The user can delete images from a catalog at will just by clicking on the image. To do that:

1. Make the image catalog active in the TOC.
2. Open the image toolbar if it is not already opened. To do this, select **Edit Model | Editing Tools** or click the button or use the popup menu.
3. Press . The application is then waiting for the user to click on the image to be removed.
4. Click on the image to be removed.
5. The image is removed from the catalog. If the image is the only image in the catalog, the catalog is also removed from the table of contents.

4.9 OVERVIEW MAP



The Overview Map allows you to see where in terms of the overall system the main network map is currently focused. This zoom area is depicted by the rectangular boundary displayed on the Overview Map. As you drag or resize this rectangle to another position the scenario within the main map will follow suit and vice versa, zooming or panning the main scenario will cause the overview map to depict the area the user is working on. The Overview Map can be toggled on and off by selecting **Scenario | Overview Map / Close OverView Map** respectively. Alternatively, click the **Close** button on the overview map to close the view.

NOTE: As the overview map is actually a *view* document, all the tools of this document may apply to this general view of the network (zoom, pan, etc.).

CHAPTER 5. IMPORTING A NETWORK TO THE SCENARIO

Importing a water distribution network CAD file appears to be a very important part of the model-making process, because generally, most of the companies involved with the management of water distribution utilities have their information diversified (CAD drawings, GIS coverages, etc), and in addition, the structure of the data source is in most cases incompatible with the configuration pursued. Hence, import tools allow to capture and convert the majority of most commonly used formats to a unique structure. This is possible with the tools that a GIS offers along with customized built-in import applications. Because not only is it a matter of capturing and redirecting data, but also a matter of debugging and adjusting all database records, additional checking modules have been developed in GISRed for that purpose using the powerful spatial analysis options of ArcView GIS.

➢ To **import a network**:

1. Make the **Link Theme active**.
2. Select **Scenario | Import (Add)** or alternatively click the button .

The following dialog will appear:

Fig. 39 Selecting the type of data to import.

3. Select the **type of data** you want to import (or add to your current network).

5.1 CAD

Use this option to **import CAD files** to your current scenario.

ArcView supports MicroStation design (.dgn) files and two kinds of AutoCAD drawing files, .dwg files (Windows only) and .dxf files (Drawing Interchange files). Before you use CAD drawings in ArcView you must first load the CAD Reader extension.

CAD Drawings typically store a variety of entity types on layers. For example, a single layer may contain graphics representing area features such as buildings, linear features such as roads, and point features such as trees and telephone poles. CAD packages generally do not restrict the way you store these entities on layers.
ArcView themes, on the other hand, draw only one class of feature in a theme. For example, a polygon theme will draw only area features and so on. Each CAD drawing entity is brought into an ArcView theme as one of the four feature classes.

Theme type	AutoCAD entities	Microstation elements
Line theme	Line, Arc, Circle, Polyline, Solid, Trace, 3DFace	Line, Line String, Complex Shape, Complex Chain, Ellipses, Arc
Point theme	Point, Shape, Insert (Block's insertion point)	Point, Cell (Cell's insertion point)Note: No shared cell support
Polygon theme	Circle, Solid, 3Dface, Closed Polylines, and Open polylines with start and end points snapped together	Ellipses, Shapes, Complex Shapes, Complex Chains with start and end points snapped together
Annotation theme	Text	Text

A drawing or a layer that does not have any of the above entity types is treated as an empty drawing or an empty layer by ArcView. Please note that the drawing or a layer may not truly be empty because it may contain some entities that aren't supported by ArcView, such as an AutoCAD's Dimension entity.

NOTE: *ArcView supports drawing files generated by AutoCAD up to Release 2000.*

➢ To **import a CAD file**:

1. Select the **CAD Layers** importation type. The dialog of the next figure will be opened.

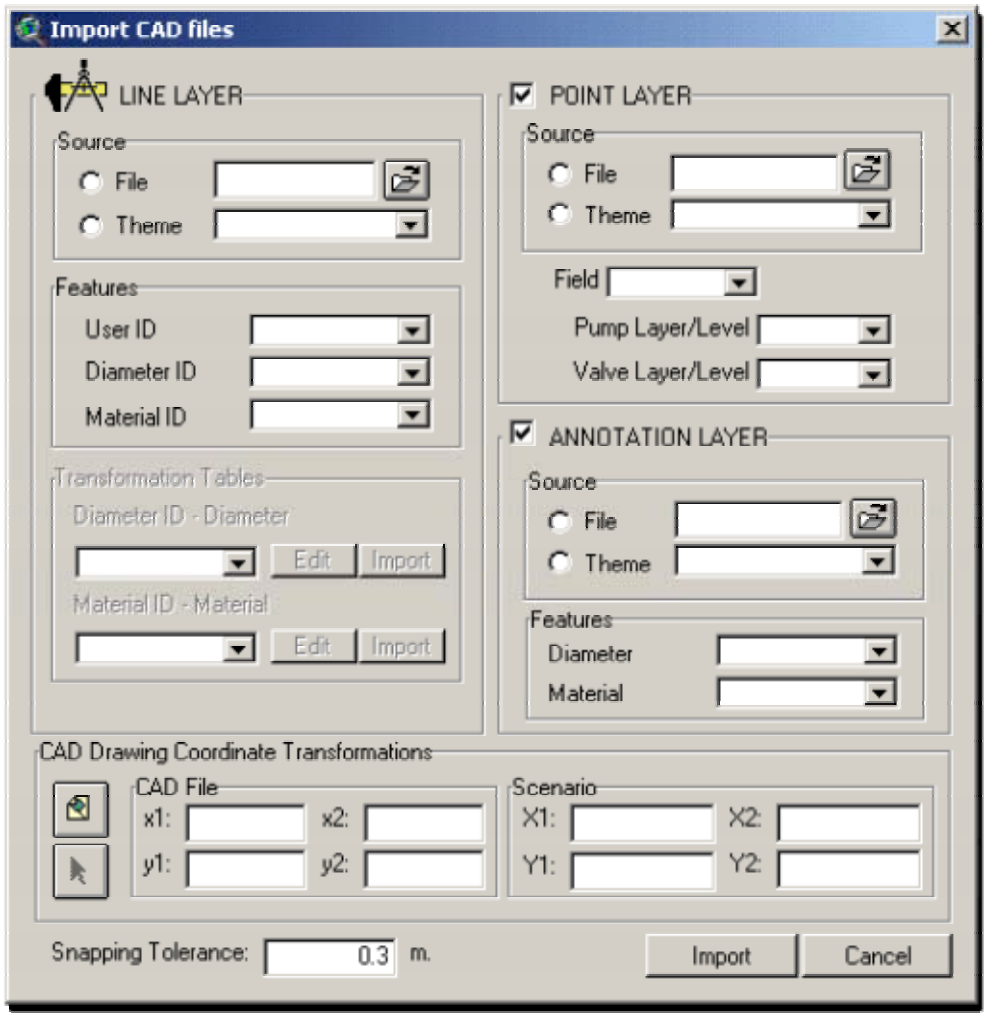


Fig. 40 CAD Importation Dialog.

2. You can select either a **Polyline/line CAD Theme** in the TOC of the current scenario (‘*Theme’ radio button*) or look for a **CAD file on disk** (‘*File’ radio button*). If the user chooses to import a CAD file that is in the TOC (‘Theme’ option), all the polyline/line CAD themes will be filtered and showed in the combo box (*Note that when a CAD file is added into ArcView, each CAD drawing entity is brought as one of the four classes aforementioned; it is recommendable that you change the name of the themes that ArcView opens in the TOC, since they share the same name and it is impossible to make the difference between them in the dialog combo box*). Once the kind of source to import has been selected, the fields of the associated table will be displayed in the next combo boxes (*Features* section).
3. Choose the **features of the associated cad theme table** to import along with the shapes. Select **NONE** if you do not want to import information for that field.
4. Select the **transformation tables** that contain the relation between Diameter ID’s - Diameter and Material ID’s - Material. [**NOTE**: *This option is still not available.*]
5. Toggle on the **Annotation Layer** checkbox if you want to capture the labels associated to each pipe (*Diameter and/or Material*). This option is useful to import the labels that are normally put on the cad lines when digitizing a network. If you choose to import annotations, select either the **Text CAD Theme** in the TOC of the current scenario (‘*Theme’ radio button*) or look for a **file on disk** (‘*File’ radio button*).

6. Enter a value for the **snapping tolerance** (*this value is 0.3 m by default*). This is really the tolerance used to check the connectivity between pipes every time a new cad line/polyline is imported.
7. The **POINT LAYER** option is *still not available*.
8. Press **Import** to start the process.

NOTE: *Remember that before starting with the import process, you can always make a selection in the CAD line/polyline theme (only applies to CAD drawing themes in the TOC) so that you will only import those selected lines/polylines.*

5.1.1 CAD DRAWING COORDINATE TRANSFORMATIONS

When you use CAD drawing files, there's no guarantee that the features will be in the same coordinate system as the data in other themes you might be displaying on your map. Some CAD drawings may be digitized in actual ground coordinates and some may be digitized in page units such as inches. In such cases, you can apply a coordinate transformation to your CAD drawing themes to ensure that they register with your themes.

On the one hand, ArcView supports either a one-point or two-point transformations. A one-point transformation simply shifts the CAD drawing theme to a new location in geographic space. A two-point transformation uses a transformation matrix that applies a coordinate offset, scale, and rotation uniformly to all coordinates read from the drawing source. The coordinate transformations are applied using the world file.

What is a world file?
The world file is a text file (with the extension .WLD) containing one or two pairs of X,Y coordinates. The first pair of X,Y coordinates is the actual X,Y location of any known control point in your drawing file. The second pair of X,Y coordinates is a new location in geographic space where you would want the CAD drawing control point to be in ArcView.

World file format
The world file format is shown below:

```
<X,Y location in CAD drawing> <space> <X,Y location in geographic space>
```

NOTES

- *The world file must have the .WLD extension and cannot have more than two lines in the above format.*
- *ArcView automatically applies the world file when it finds a valid file name with the same prefix as the drawing file name in the same sub-directory. For example, suppose you've stored sample.dwg and sample.wld files in the same sub-directory. When you add themes from sample.dwg, ArcView automatically uses the sample.wld file for coordinate transformations.*

On the other hand, the CAD importation dialog allows the user to define directly the two pairs of X,Y coordinates just by clicking on the corresponding points of the CAD drawing and the scenario.

➢ To **apply a coordinate transformation to a theme based on a CAD drawing**:

1. Once the line layer has been selected, press on .
2. A new view will be opened showing the CAD drawing.
3. Now, press the **arrow button**.
4. Click the **first** point on the **CAD drawing** (view). *Note how the coordinates of the point are written in the x1, y1 fields*.

5. Click the **first** point on the **Scenario**. *Note how the coordinates of the point are written in the X1, Y1 fields.*
6. Click the **second** point on the **CAD drawing** (view).
7. Click the **second** point on the **Scenario**.
8. When all the points have been defined, the user can close the CAD File Preview window and go on with the import process.

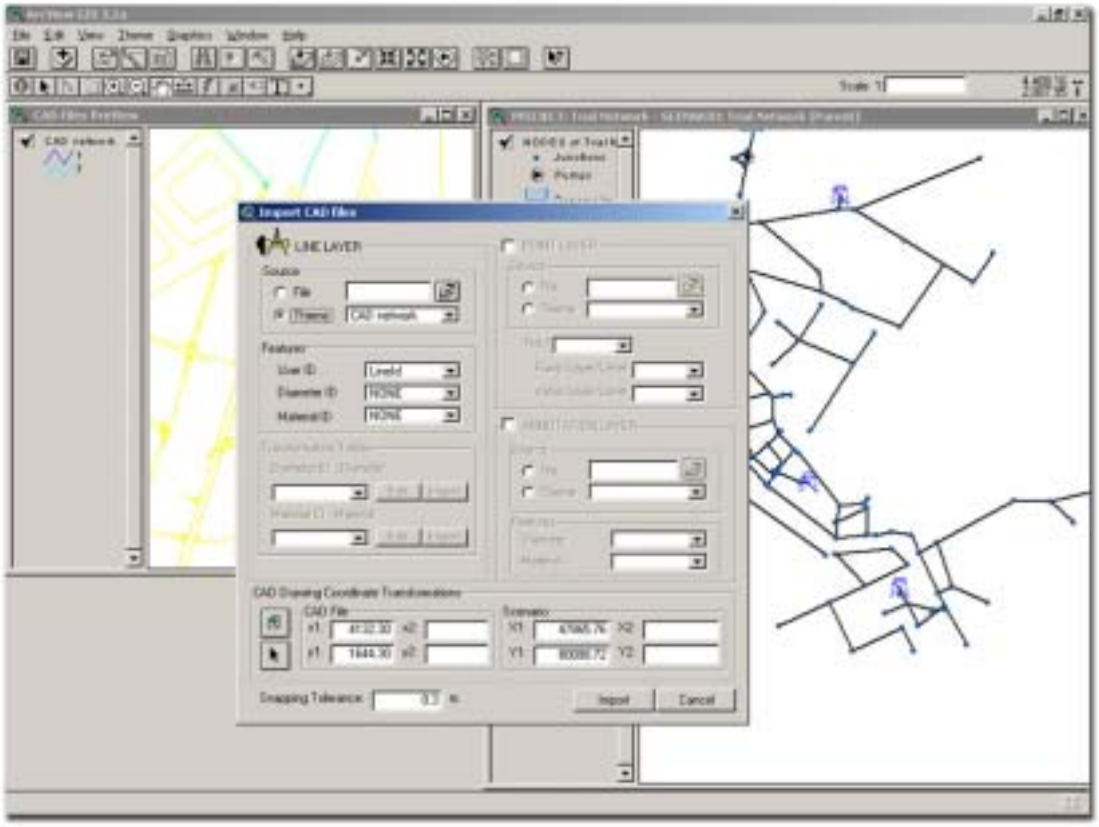


Fig. 41 Applying a Coordinate Transformation.

5.2 SHAPEFILES

Use this option to import an EPANET input file to your current scenario

➢ To **import an ArcView Shapefile**:

1. Select the **SHAPE Files** importation type. The dialog of the next figure will be opened.
2. You can select either a **Polyline Theme** in the TOC of the current scenario (‘*Theme’ radio button*) or look for a **shapefile on disk** (‘*File’ radio button*). If the user chooses to import a shapefile that is in the TOC (‘Theme’ option), all the polyline themes will be filtered and showed in the combo box. Once the kind of source to import has been selected, the fields of the associated table will be displayed in the next combo boxes (Fields section).

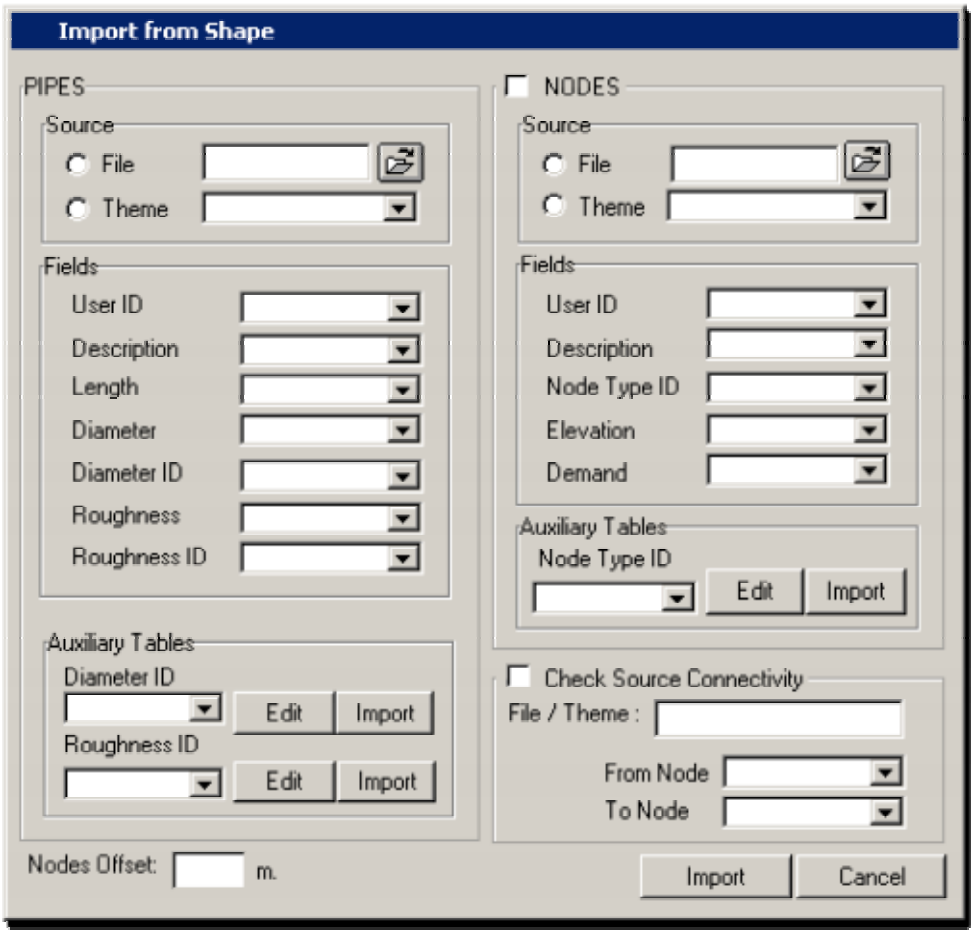


Fig. 42 Shapefile Importation Dialog.

3. Choose the **fields of the associated link table** to import along with the shapes. These fields correspond to the properties specified in the next framed section. Choose **NONE** if you do not want to import that field.
4. Select the **auxiliary tables** that contain the relation between Diameter ID’s and Diameter values. Same for the roughness. [**NOTE**: *This option is still not available.*]

5. Enter a value for the **nodes offset** (*this value is 0.3 m by default*). This is really the tolerance used to check the connectivity between pipes every time a new pipe (polyline/line) is imported.
6. Toggle on the **NODES** checkbox if you you have a node shapefile which to compare against. This is used to compare the end nodes that the application creates every time a new pipe is imported with a shapefile that contains a set of nodes with certain characteristics.
7. Choose the **fields of the associated node table** to import. These fields correspond to the properties specified in the next framed section. Choose **NONE** if you do not want to import that field. These properties are imported just when a node from the shapefile matches geographically (within the specified tolerance) with the nodes that the application create during the link importation process. If the '*existing*' and '*read*' node match geographically, the '*existing*' node gets all the properties of the '*read*' node.
8. **NOTE**: The node '*auxiliary tables'* section is not used anymore (.
9. Toggle on the **Check Source Connectivity** to check the connectivity against the selected link shapefile or link theme. Specify the *FROM NODE* and *TO NODE* fields of the associated table. This is used to find out topology inconsistencies in the original information that is being imported.
10. Press **Import** to start the process.

NOTES:

- *Once the process is finished, a summary window will appear specifying the number of pipes imported, the number of nodes created, and the number of errors detected.*
- *If during the importation of pipes the application detects that one pipe already exists, the user will be asked what to do with the actual pipe, that is, either replace it or not.*
- *After the importation process, if errors or inconsistencies were found, an error theme will be added to the TOC of the scenario.*

5.3 ERROR THEMES AND NAVIGATOR

When importing either CAD files or Shapefiles, the application looks for errors that the original files might have (mainly inconsistencies in the layout). After the importation, if any error was detected, the application will add an error theme to the TOC of the scenario. It is possible to navigate from one error to another using the navigator utility.

➢ To use the Error Navigator:

1. Make the **Error Theme active**.
2. Select **Edit Model | Error Navigator**.
3. A new view will be opened. This view presents four buttons that can be used to move from one error to another, close the view or remove an error.

NOTE: *the user can navigate through the errors and fix them in the scenario using all the available tools.*

Still Not Available

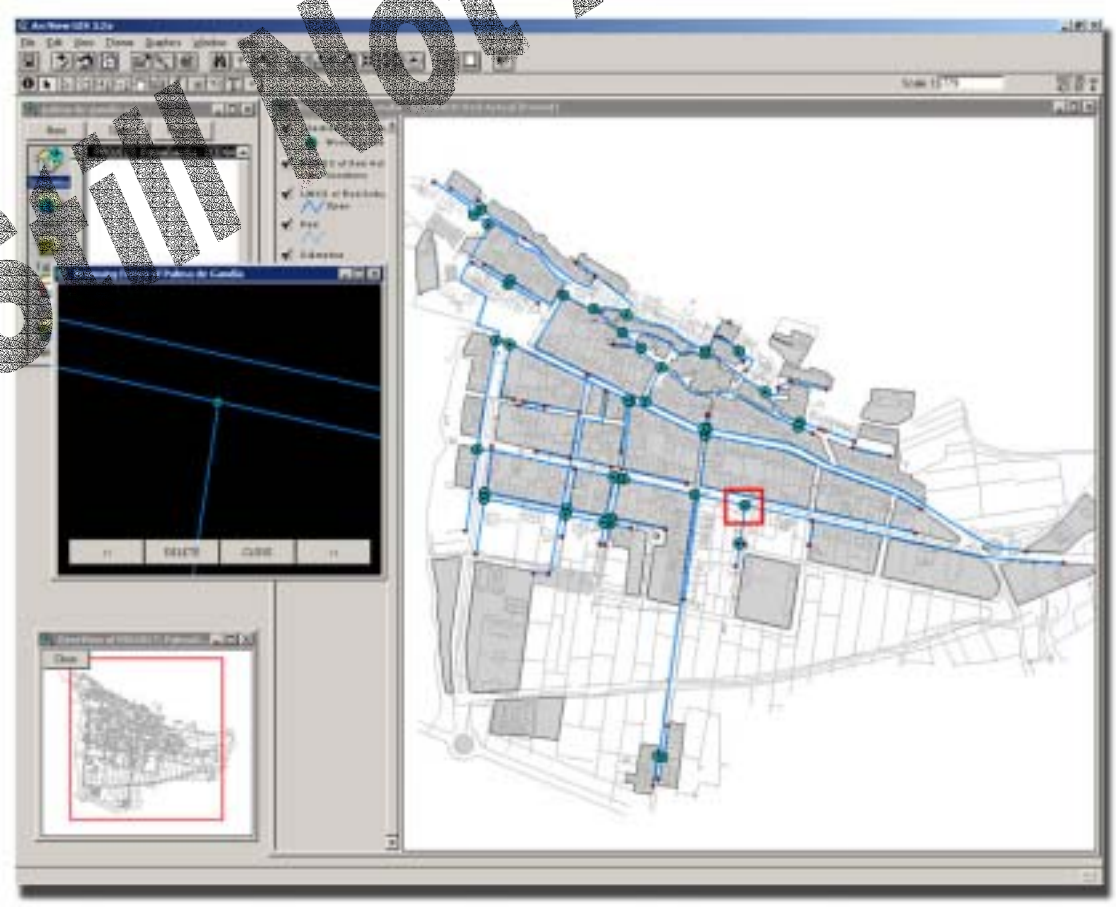


Fig. 43 Error Navigator.

5.4 GIS COVERAGES

This part is still not developed. It will allow the user to import ArcINFO coverages into the scenario.

5.5 EPANET INPUT FILES

Use this option to import an EPANET input file to your current scenario. It is possible to import Epanet input files from the version 1.0 and 2.0 (*just *.inp files*). If an Epanet input file v1.0 is to be imported the application will look for the reference to the *.map file (*coordinates file*) in the input file (**[MAP]** *section*); if it is not found, the user will be prompted to look for that file manually.

NOTE: *Only pipes and junctions will be imported.*

➢ To import an **EPANET input file**:

11. Select the **EPANET Input Files** importation type.
12. **Enter the path** of the INP file or press on **Search** to look for it manually.
13. Press **Import**.

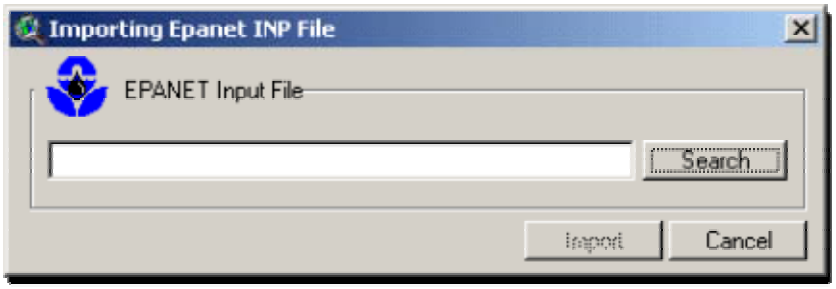


Fig. 44 Importing an EPANET input file.

5.6 GISRED PROJECT

Use this option to import an GISRed project to a new scenario. This is useful when the user wants to import an old version of an GISRed project. The application looks for certain basic fields in the old database and adapts the data into the structure of the new version.

➢ To **import an GISRed** project:

1. Select the **GISRed Project importation** type.
2. Fill in all the fields to create a new project and the parent scenario.
3. Press **OK** to import the whole project.

CHAPTER 6. EDITING NETWORK PROPERTIES

There are a series of property editors associated to each network element that can appear on the Network Map (Junctions, Reservoirs, Tanks, Pipes, Pumps and Valves). To edit one of these network elements, make active the corresponding theme associated to the element (pipe theme / node theme) and click the '***Model Properties***' tool . Now, select the object on the scenario. The properties associated with each element are described below. To see the properties of another element, click directly on the element or use the property editor navigator.

PROPERTY EDITOR NAVIGATOR	
TOOL	**DESCRIPTION**
	FIRST ELEMENT: The editor shows the properties of the first element in the list.
	PREVIOUS ELEMENT: The editor shows the properties of the previous element in the list.
	NEXT ELEMENT: The editor shows the properties of the next element in the list.
	LAST ELEMENT: The editor shows the properties of the last element in the list.
	FIND ELEMENT: Finds an element by its name and shows its properties.

NOTE: *The unit system in which element properties are expressed in the editor, depends on the choice of units for flow rate. Using a flow rate expressed in cubic feet, gallons or acre-feet means that US units will be used for all quantities. Using a flow rate expressed in liters or cubic meters means that SI metric units will be used. Flow units are selected from the scenario's Hydraulic Options which can be accessed from the* ***Scenario*** *|* ***Analysis Options*** *or from the Project Defaults dialog which can be accessed from* ***Scenario*** *|* ***Defaults*** *menu. The units used for all properties are summarized In* APPENDIX B – UNITS OF MEASUREMENT.

6.1 JUNCTION PROPERTIES

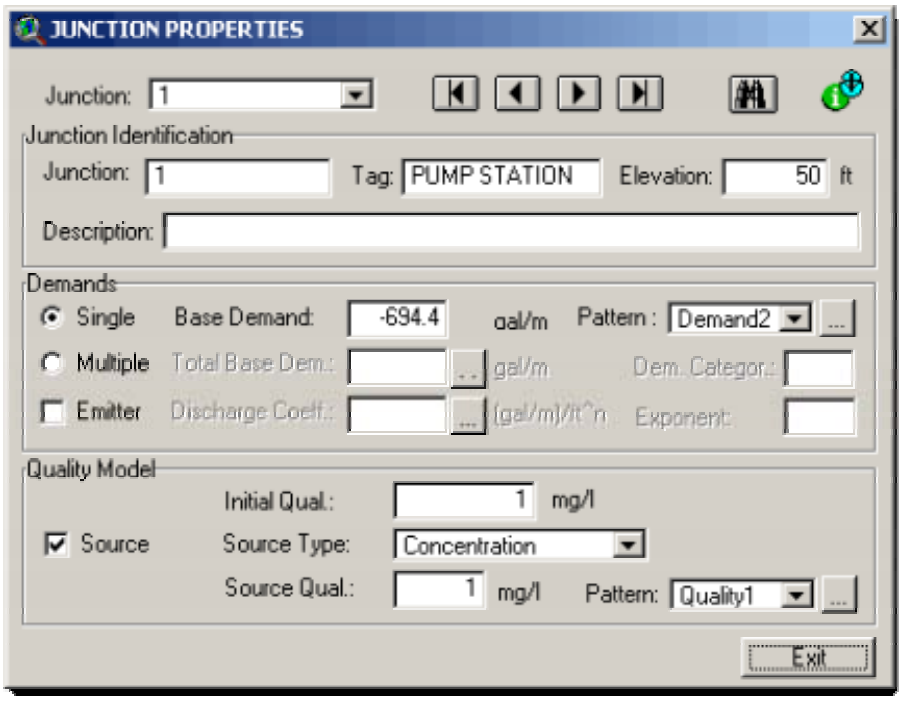


Fig. 45 Junction Properties.

PROPERTY	DESCRIPTION
JUNCTION IDENTIFICATION	
Junction	A unique label used to identify the junction. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
Description	An optional text string that describes other significant information about the junction.
Tag	An optional text string used to assign the junction to a category, such as a pressure zone.
Elevation	The elevation in feet (meters) above some common reference of the junction. This is a required property. Elevation is used only to compute pressure at the junction. It does not affect any other computed quantity.
DEMANDS	
Single: When only a single demand applies for the junction.	
Base Demand	The average or nominal demand for water by the main category of consumer at the junction, as measured in the current flow units. A negative value is used to indicate an external source of flow into the junction. If left blank then demand is assumed to be zero.
Demand Pattern	The ID label of the time pattern used to characterize time variation in demand for the main category of consumer at the junction. The pattern provides multipliers that are applied to the Base Demand to determine actual demand in a given time period. If left blank then

	the Default Time Pattern assigned in the Hydraulic Options will be used. Click the ellipsis button to bring up the Pattern Editor.
Multiple: When a number of different categories of water use apply. Ignore if only a single demand category will suffice.	
Total Base Demand	It is the total sum of all the base demands of each category. Click the ellipsis button to bring up the *Multiple Demands Editor* which will let you assign base demands and time patterns to multiple categories of users at the junction. See the editor below.
Demand Categories	Number of different categories of water users defined for the junction.
Emitter (see explanation below)	
Discharge Coefficient	Discharge coefficient for emitter (sprinkler or nozzle) placed at junction. The coefficient represents the flow (in current flow units) that occurs at a pressure drop of 1 psi (or meter). Leave blank if no emitter is present. See the Emitters explanation below for more details.
Exponent	Power to which pressure is raised when computing the flow through an emitter device. The textbook value for nozzles and sprinklers is ½. This may not apply to pipe leakage.
QUALITY MODEL	
Initial Quality	Water quality level at the junction at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
Source	The Source checkbox is used to describe the quality of source flow entering the network at a specific node. This source might represent the main treatment works, a well head or satellite treatment facility, or an unwanted contaminant intrusion. Check this property to specify a source at that junction.
Source Type	A water quality source can be designated as a concentration or booster source. ▪ A **concentration** source fixes the concentration of any external inflow entering the network, such as flow from a reservoir or from a negative demand placed at a junction. ▪ A **mass booster** source adds a fixed mass flow to that entering the node from other points in the network. ▪ A **flow paced booster** source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network. ▪ A **setpoint booster** source fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint). The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion
Source Quality	Quality of any water entering the network at this location.
Quality Pattern	ID label of time pattern used to make source quality vary with time. Leave blank if not applicable. Click the ellipsis button to bring up the Pattern Editor.

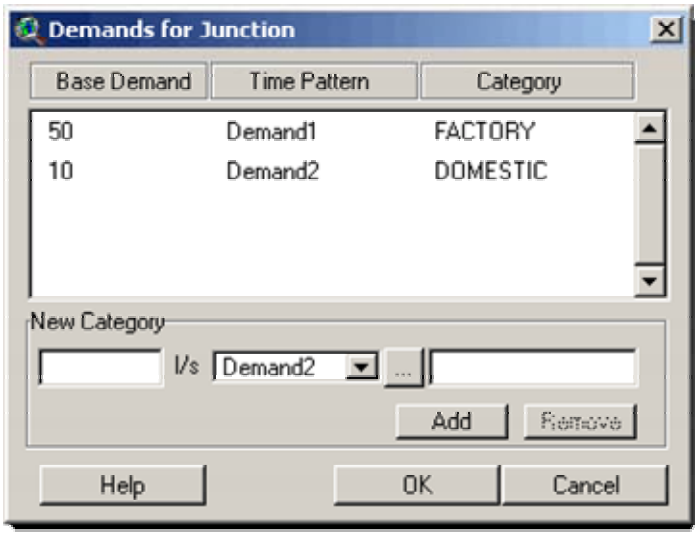
6.1.1 Emitters

Emitters are devices associated with junctions that model the flow through a nozzle or orifice that discharges to the atmosphere. The flow rate through the emitter varies as a function of the pressure available at the node:

$$q = C\ p^{\gamma}$$

where q = flow rate, p = pressure, C = discharge coefficient, and γ = pressure exponent. For nozzles and sprinkler heads γ equals 0.5 and the manufacturer usually provides the value of the discharge coefficient in units of gpm/psi$^{0.5}$ (stated as the flow through the device at a 1 psi pressure drop).

Emitters are used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated) or compute a fire flow at the junction (the flow available at some minimum residual pressure). In the latter case one would use a very high value of the discharge coefficient (e.g., 100 times the maximum flow expected) and modify the junction's elevation to include the equivalent head of the pressure target. GISRed, as in EPANET, treats emitters as a property of a junction and not as a separate network component.



6.1.2 Multiple Demand Editor

The Multiple Demand Editor is pictured in the adjoined figure. It is used to assign base demands and time patterns when there is more than one category of water user at a junction. The editor is invoked from the Junction Property Editor by clicking the ellipsis button beside the *Total Base Demand* field.

The editor is basically a listbox containing three columns. Each category of demand is entered as a new row in the listbox. The columns contain the following information:

- **Base Demand**: baseline or average demand for the category (*required*)
- **Time Pattern**: ID label of time pattern used to allow demand to vary with time (*optional*)
- **Category**: text label used to identify the demand category (*optional*)

To enter a new category, fill in at least the *Base Demand* and press **Add** (or alternatively hit *Enter* key when the focus is at the category field).

To remove a category, select a row and press **Remove**.

Note: The total demand sum of the first column (*Base Demand*) of the listbox will be showed in the **Total Base Demand** field of the Junction Property Editor.

6.2 RESERVOIR PROPERTIES

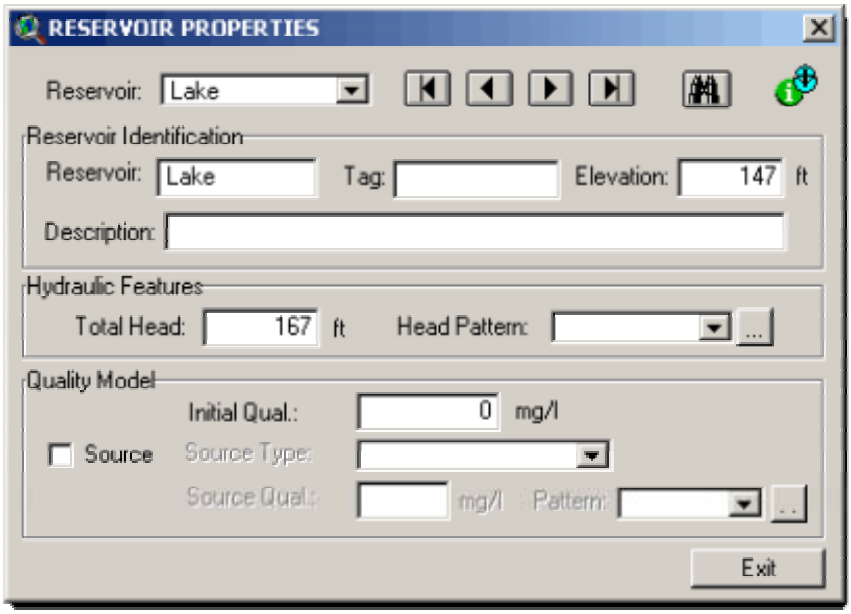


Fig. 46 Reservoir Properties.

PROPERTY	DESCRIPTION
RESERVOIR IDENTIFICATION	
Reservoir	A unique label used to identify the reservoir. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
Description	An optional text string that describes other significant information about the reservoir.
Tag	An optional text string used to assign the reservoir to a category, such as a pressure zone.
Elevation	The elevation in feet (meters) above some common reference of the reservoir. This property is optional, it gives extra information.
HYDRAULIC FEATURES	
Total Head	The hydraulic head (elevation + pressure head) of water in the reservoir in feet (meters). This is a required property.
Head Pattern	The ID label of a time pattern used to model time variation in the reservoir's head. Leave blank if none applies. This property is useful if the reservoir represents a tie-in to another system whose pressure varies with time. Click the ellipsis button to bring up the Pattern Editor.
QUALITY MODEL	
Initial Quality	Water quality level at the reservoir at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
Source	The Source checkbox is used to describe the quality of source flow entering the network at a specific node. This source might represent the main treatment works, a well head or satellite

	treatment facility, or an unwanted contaminant intrusion. Check this property to specify a source at the reservoir.
Source Type	A water quality source can be designated as a concentration or booster source. ▪ A **concentration** source fixes the concentration of any external inflow entering the network, such as flow from a reservoir or from a negative demand placed at a junction. ▪ A **mass booster** source adds a fixed mass flow to that entering the node from other points in the network. ▪ A **flow paced booster** source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network. ▪ A **setpoint booster** source fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint). The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion
Source Quality	Quality of any water entering the network at this location.
Quality Pattern	ID label of time pattern used to make source quality vary with time. Leave blank if not applicable. Click the ellipsis button to bring up the Pattern Editor.

6.3 TANK PROPERTIES

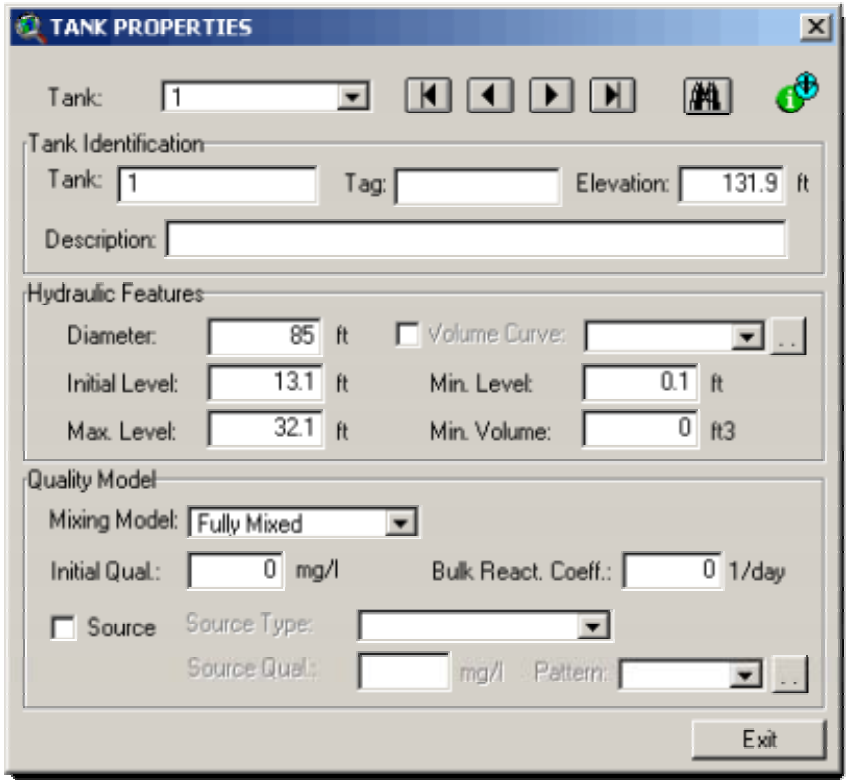


Fig. 47 Tank Properties.

PROPERTY	DESCRIPTION
TANK IDENTIFICATION	
Tank	A unique label used to identify the tank. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other node. This is a required property.
Description	An optional text string that describes other significant information about the tank.
Tag	An optional text string used to assign the tank to a category, such as a pressure zone.
Elevation	Elevation above a common datum in feet (meters) of the bottom shell of the tank. This is a required property.
HYDRAULIC FEATURES	
Diameter	The diameter of the tank in feet (meters). For cylindrical tanks this is the actual diameter. For square or rectangular tanks it can be an equivalent diameter equal to 1.128 times the square root of the cross-sectional area. For tanks whose geometry will be described by a curve (see below) it can be set to any value. This is a required property.
Volume Curve	The ID label of a curve used to describe the relation between tank volume and water level. Toggle the checkbox off if the curve is not required. If no value is supplied then the tank is assumed to be cylindrical. Click the ellipsis button to bring up the Curve Editor.

Initial Level	Height in feet (meters) of the water surface above the bottom elevation of the tank at the start of the simulation. This is a required property.
Minimum Level	Minimum height in feet (meters) of the water surface above the bottom elevation that will be maintained. The tank will not be allowed to drop below this level. This is a required property.
Maximum Level	Maximum height in feet (meters) of the water surface above the bottom elevation that will be maintained. The tank will not be allowed to rise above this level. This is a required property.
Minimum Volume	The volume of water in the tank when it is at its minimum level, in cubic feet (cubic meters). This is an optional property, useful mainly for describing the bottom geometry of non-cylindrical tanks where a full volume versus depth curve will not be supplied (see below).
QUALITY MODEL	
Mixing Model	The type of water quality mixing that occurs within the tank. The choices include: ▪ **MIXED** (fully mixed), ▪ **2COMP** (two-compartment mixing), ▪ **FIFO** (first-in-first-out plug flow), ▪ **LIFO** (last-in-first-out plug flow). See the 'Mixing in Storage Tanks' topic below for more information.
Mixing Fraction	The fraction of the tank's total volume that comprises the inlet-outlet compartment of the two-compartment (2COMP) mixing model. This property is only specified when the 2COMP mixing model is chosen. Can be left blank if another type of mixing model is employed.
Initial Quality	Water quality level at the tank at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
Bulk Reaction Coefficient	The bulk reaction coefficient for chemical reactions in the tank. Time units are 1/days. Use a positive value for growth reactions and a negative value for decay. Leave blank if the Global Bulk reaction coefficient specified in the project's Reactions Options will apply. See 'Water Quality Reactions' below for more information.
Source	The Source checkbox is used to describe the quality of source flow entering the network at a specific node. This source might represent the main treatment works, a well head or satellite treatment facility, or an unwanted contaminant intrusion. Check this property to specify a source at the reservoir.
Source Type	A water quality source can be designated as a concentration or booster source. ▪ A **concentration** source fixes the concentration of any external inflow entering the network, such as flow from a reservoir or from a negative demand placed at a junction. ▪ A **mass booster** source adds a fixed mass flow to that entering the node from other points in the network. ▪ A **flow paced booster** source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network. ▪ A **setpoint booster** source fixes the concentration of any flow leaving the node (as long as the concentration

	resulting from all inflow to the node is below the setpoint). The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion
Source Quality	Quality of any water entering the network at this location.
Quality Pattern	ID label of time pattern used to make source quality vary with time. Leave blank if not applicable. Click the ellipsis button to bring up the Pattern Editor.

6.3.1 Mixing in Storage Tanks

GISRed can use four different types of models to characterize mixing within storage tanks as illustrated in the next figure:

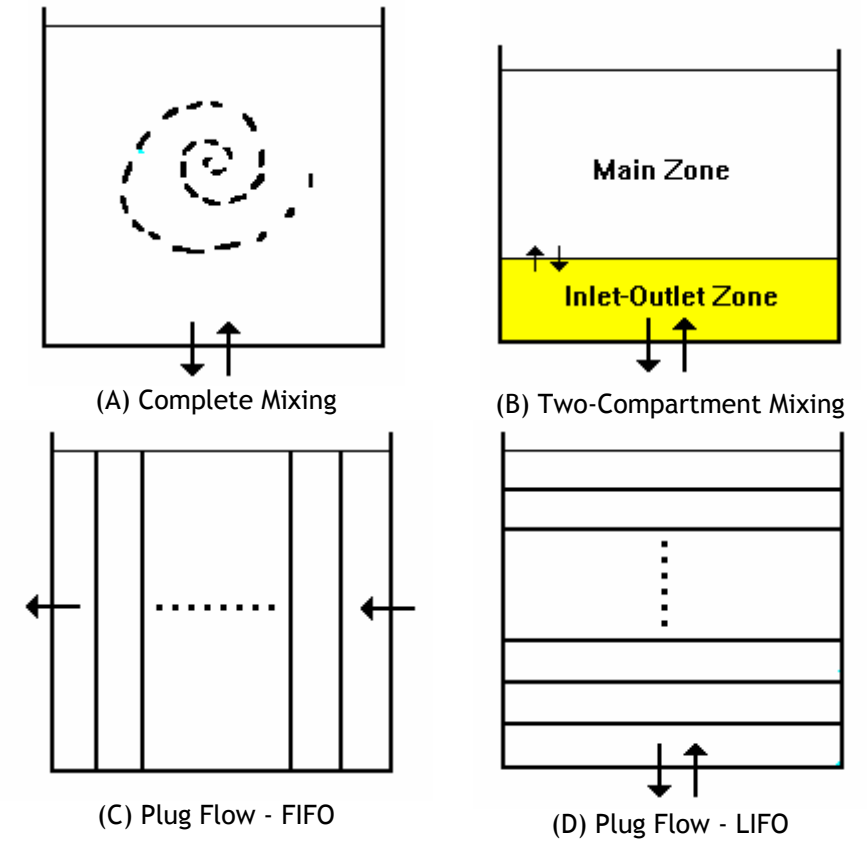
Complete Mixing

Two-Compartment Mixing

FIFO (First in first out) Plug Flow

LIFO (Last in first out) Plug Flow

Different models can be used with different tanks within a network.



(A) Complete Mixing

(B) Two-Compartment Mixing

(C) Plug Flow - FIFO

(D) Plug Flow - LIFO

The **Complete Mixing model** (Figure (a)) assumes that all water that enters a tank is instantaneously and completely mixed with the water already in the tank. It is the simplest

form of mixing behavior to assume, requires no extra parameters to describe it, and seems to apply quite well to a large number of facilities that operate in fill-and-draw fashion.

The **Two-Compartment Mixing model** (Figure (b)) divides the available storage volume in a tank into two compartments, both of which are assumed completely mixed. The inlet/outlet pipes of the tank are assumed to be located in the first compartment. New water that enters the tank mixes with the water in the first compartment. If this compartment is full, then it sends its overflow to the second compartment where it completely mixes with the water already stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of water from the second compartment to make up the difference. The first compartment is capable of simulating short-circuiting between inflow and outflow while the second compartment can represent dead zones. The user must supply a single parameter, which is the fraction of the total tank volume devoted to the first compartment.

The **FIFO Plug Flow model** (Figure (c)) assumes that there is no mixing of water at all during its residence time in a tank. Water parcels move through the tank in a segregated fashion where the first parcel to enter is also the first to leave. Physically speaking, this model is most appropriate for baffled tanks that operate with simultaneous inflow and outflow. There are no additional parameters needed to describe this mixing model.

The **LIFO Plug Flow model** (Figure (d)) also assumes that there is no mixing between parcels of water that enter a tank. However in contrast to FIFO Plug Flow, the water parcels stack up one on top of another, where water enters and leaves the tank on the bottom. This type of model might apply to a tall, narrow standpipe with an inlet/outlet pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided.

6.4 PIPE PROPERTIES

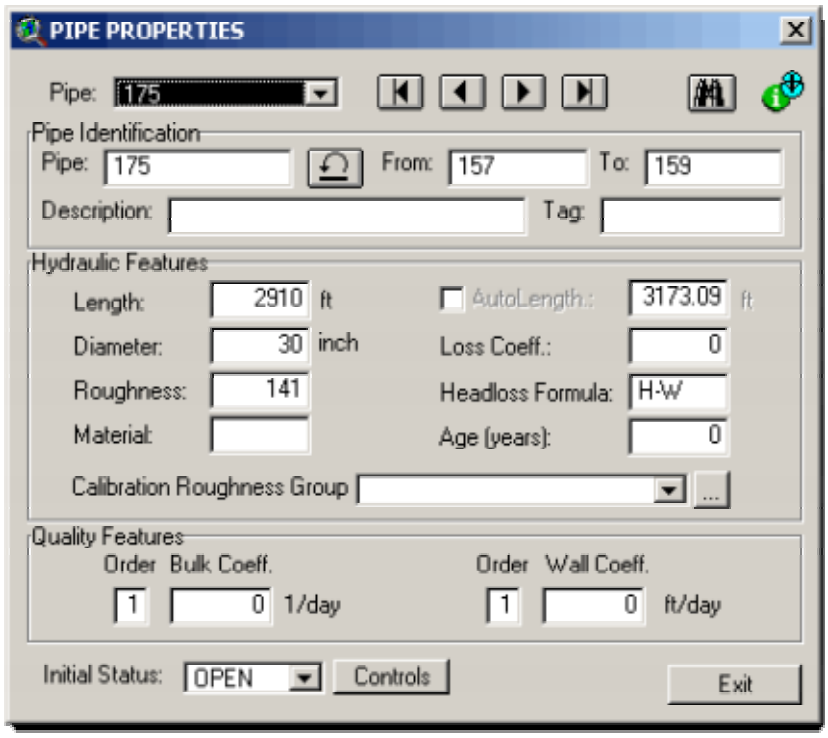


Fig. 48 Pipe Properties.

PROPERTY	DESCRIPTION
PIPE IDENTIFICATION	
Pipe	A unique label used to identify the pipe. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Flip button	Reverses the order of vertices in a Pipe.
Description	An optional text string that describes other significant information about the pipe.
Tag	An optional text string used to assign the pipe to a category, perhaps one based on age or material.
From	Start node. The ID of the node where the pipe begins. This is a non editable property.
To	End Node. The ID of the node where the pipe ends. This is a non editable property.
HYDRAULIC FEATURES	
Length	The actual length of the pipe in feet (meters). This is a required property when the AutoLength checkbox is not on.
AutoLength	The automatic length of the pipe in map units. This property is calculated by GISRed anyway, but it is only used if the user turns on the checkbox. *Note*: Pipe lengths are automatically computed as pipes are added or repositioned on the network map if the Auto-Length setting is

	turned on. To toggle this setting On/Off: - Select **Scenario | Defaults** and edit the Auto-Length field on the Properties page of the Defaults dialog form. Be sure to provide meaningful dimensions for the scenario map units (View | Properties) before using the Auto-Length feature.
Diameter	The pipe diameter in inches (mm). This is a required property.
Loss Coefficient	Unitless minor loss coefficient associated with bends, fittings, etc. Assumed 0 if left blank.
Roughness	The roughness coefficient of the pipe. It is unitless for Hazen-Williams or Chezy-Manning roughness and has units of millifeet (mm) for Darcy-Weisbach roughness. This is a required property.
Headloss Formula	This field indicates which headloss formula is being used. This is a non editable property. To change this property, use the *Defaults Dialog* or the *Hydraulic Options*.
Material	The pipe material. This is just an information field. This is not a required property. It is useful in operations such as calibration or node deletion.
Age	The pipe age. This is just an information field. This is not a required property. It is useful in the process of calibration grouping.
Calibration Roughness Group	This field indicates which calibration roughness group the pipe belongs to. This is not a required property. Leave blank if the pipe does not belong to any group or calibration is not being perfomed. Click the ellipsis button to bring up the Roughness Grouping Editor. See the Calibration Section for more details.
QUALITY FEATURES	
Bulk Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored. This is a non editable property. To change the value select *Scenario | Analysis Options | Quality-Reactions.*
Bulk Coefficient	The bulk reaction coefficient for the pipe. Time units are 1/days. Use a positive value for growth and a negative value for decay. Leave blank if the Global Bulk reaction coefficient from the project's Reaction Options will apply. See '*Water Quality Reactions*' below for more information.
Wall Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Choices are (1) for first-order reactions or (0) for constant rate reactions. If no global or pipe-specific wall reaction coefficients are assigned then this option is ignored. This is a non editable property. To change the value select *Scenario | Analysis Options | Quality-Reactions.*
Wall Coefficient	The wall reaction coefficient for the pipe. Time units are 1/days. Use a positive value for growth and a negative value for decay. Leave blank if the Global Wall reaction coefficient from the project's Reactions Options will apply. See '*Water Quality Reactions*' below for more information.
OTHER	
Initial Status	Determines whether the pipe is initially open, closed, or contains a check valve. If a check valve is specified then the flow direction in the pipe will always be from the Start node to the End node.

Controls	By pressing this button the application checks if there are control rules associated to the pipe. If so, an information window is opened specifying the control rules and highlighting the nodes the pipe is dependant upon.

6.4.1 Water Quality Reactions

GISRed by means of the EPANET engine, can track the growth or decay of a substance by reaction as it travels through a distribution system. In order to do this it needs to know the rate at which the substance reacts and how this rate might depend on substance concentration. Reactions can occur both within the bulk flow and with material along the pipe wall. This is illustrated in Fig. 49. In this example free chlorine (HOCl) is shown reacting with natural organic matter (NOM) in the bulk phase and is also transported through a boundary layer at the pipe wall to oxidize iron (Fe) released from pipe wall corrosion. Bulk fluid reactions can also occur within tanks. GISRed allows a modeler to treat these two reaction zones separately.

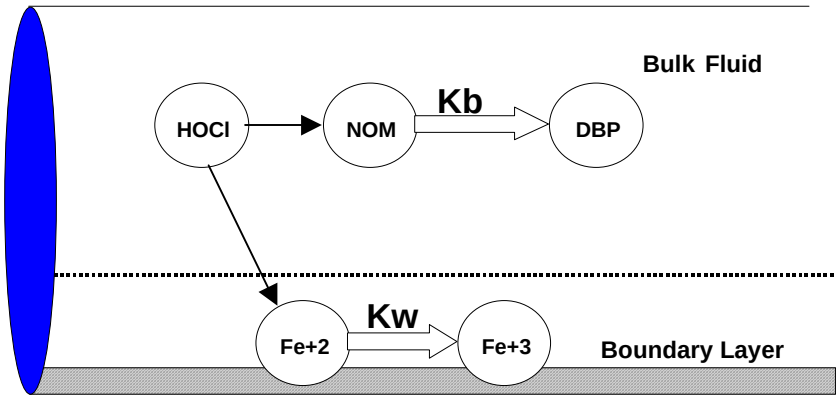


Fig. 49 Reactions zones within a pipe.

6.4.1.1 Bulk Reactions

EPANET models reactions occurring in the bulk flow with n-th order kinetics, where the instantaneous rate of reaction (R in mass/volume/time) is assumed to be concentration-dependent according to

$$R = K_b C^n$$

Here K_b = a bulk reaction rate coefficient, C = reactant concentration (mass/volume), and n = a reaction order. K_b has units of concentration raised to the (1-n) power divided by time. It is positive for growth reactions and negative for decay reactions.

EPANET can also consider reactions where a limiting concentration exists on the ultimate growth or loss of the substance. In this case the rate expression becomes

$$R = K_b (C_L - C) C^{(n-1)} \qquad \text{for } n > 0,\ K_b > 0$$

$$R = K_b (C - C_L) C^{(n-1)} \qquad \text{for } n > 0,\ K_b < 0$$

where C_L = the limiting concentration. Thus there are three parameters (K_b, C_L, and n) that are used to characterize bulk reaction rates. Some special cases of well-known kinetic models include the following:

Model	*Parameters*	*Examples*
First-Order Decay	$C_L = 0$, $K_b < 0$, $n = 1$	Chlorine
First-Order Saturation Growth	$C_L > 0$, $K_b > 0$, $n = 1$	Trihalomethanes
Zero-Order Kinetics	$C_L = 0$, $K_b <> 0$, $n = 0$	Water Age
No Reaction	$C_L = 0$, $K_b = 0$	Fluoride Tracer

The K_b for first-order reactions can be estimated by placing a sample of water in a series of non-reacting glass bottles and analyzing the contents of each bottle at different points in time. If the reaction is first-order, then plotting the natural log (C_t/Co) against time should result in a straight line, where C_t is concentration at time t and Co is concentration at time zero. K_b would then be estimated as the slope of this line.

Bulk reaction coefficients usually increase with increasing temperature. Running multiple bottle tests at different temperatures will provide more accurate assessment of how the rate coefficient varies with temperature.

6.4.1.2 Wall Reactions

The rate of water quality reactions occurring at or near the pipe wall can be considered to be dependent on the concentration in the bulk flow by using an expression of the form

$$R = (A/V)K_w C^n$$

where K_w = a wall reaction rate coefficient and (A/V) = the surface area per unit volume within a pipe (equal to 4 divided by the pipe diameter). The latter term converts the mass reacting per unit of wall area to a per unit volume basis. EPANET limits the choice of wall reaction order to either 0 or 1, so that the units of K_w are either mass/area/time or length/time, respectively. As with K_b, K_w must be supplied to the program by the modeler. First-order K_w values can range anywhere from 0 to as much as 5 ft/day.

K_w should be adjusted to account for any mass transfer limitations in moving reactants and products between the bulk flow and the wall. EPANET does this automatically, basing the adjustment on the molecular diffusivity of the substance being modeled and on the flow's Reynolds number. Setting the molecular diffusivity to zero will cause mass transfer effects to be ignored.

The wall reaction coefficient can depend on temperature and can also be correlated to pipe age and material. It is well known that as metal pipes age their roughness tends to increase due to encrustation and tuberculation of corrosion products on the pipe walls. This increase in roughness produces a lower Hazen-Williams C-factor or a higher Darcy-Weisbach roughness coefficient, resulting in greater frictional head loss in flow through the pipe.

There is some evidence to suggest that the same processes that increase a pipe's roughness with age also tend to increase the reactivity of its wall with some chemical species, particularly chlorine and other disinfectants. EPANET can make each pipe's K_w be a function of the coefficient used to describe its roughness. A different function applies depending on the formula used to compute headloss through the pipe:

Headloss Formula	Wall Reaction Formula
Hazen-Williams	$K_w = F / C$
Darcy-Weisbach	$K_w = -F / \log(e/d)$
Chezy-Manning	$K_w = F\ n$

where *C* = Hazen-Williams C-factor, *e* = Darcy-Weisbach roughness, *d* = pipe diameter, *n* = Manning roughness coefficient, and *F* = wall reaction - pipe roughness coefficient The coefficient F must be developed from site-specific field measurements and will have a different meaning depending on which head loss equation is used. The advantage of using this approach is that it requires only a single parameter, *F*, to allow wall reaction coefficients to vary throughout the network in a physically meaningful way.

6.5 PUMP PROPERTIES

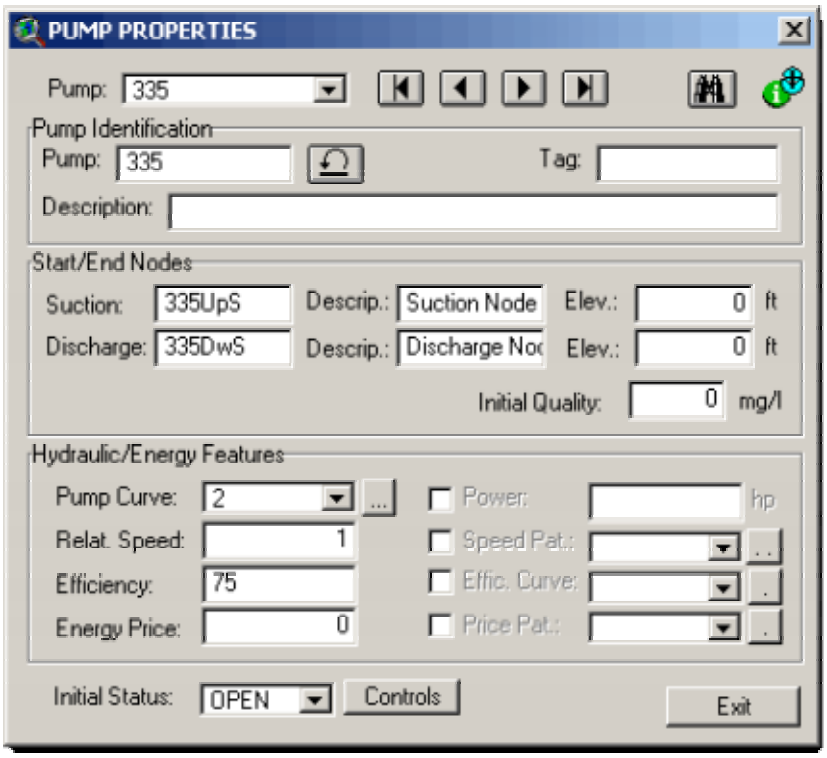


Fig. 50 Pump Properties.

PROPERTY	DESCRIPTION
PUMP IDENTIFICATION	
Pump	A unique label used to identify the pump. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Flip button	Reverses the direction of the pump.
Description	An optional text string that describes other significant information about the pump.
Tag	An optional text string used to assign the pump to a category, perhaps based on age, size or location..
START/END NODES	
Suction	Start node. The ID of the node on the suction side of the pump. This is a non editable property. By default the Suction Node is named as the Pump ID but adding the 'UpS' suffix (meaning upstream).
Description (Suction node)	An optional text string that describes other significant information about the pump suction node.
Elevation (Suction node)	The elevation in feet (meters) above some common reference of the pump suction. This is a required property. Elevation is used only to compute pressure at that point. It does not affect any other computed quantity.
Discharge	End Node. The ID of the node on the suction side of the pump. This is a non editable property. By default the Discharge Node is named

	as the Pump ID but adding the 'DwS' suffix (meaning downstream).
Description (Discharge node)	An optional text string that describes other significant information about the pump discharge node.
Elevation (Discharge node)	The elevation in feet (meters) above some common reference of the pump discharge. This is a required property. Elevation is used only to compute pressure at that point. It does not affect any other computed quantity.
Initial Quality	Water quality level at the suction and discharge nodes at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
HYDRAULIC AND ENERGY FEATURES	
Pump Curve	The ID label of the pump curve used to describe the relationship between the head delivered by the pump and the flow through the pump. It is left blank if the pump will be a constant energy pump (power checkbox is selected). Click the ellipsis button to bring up the Curve Editor.
Power	The power supplied by the pump in horsepower (kw). Assumes that the pump supplies the same amount of energy no matter what the flow is. Do not check this field if a pump curve will be used instead. Use when pump curve information is not available.
Relative Speed	The relative speed setting of the pump (unitless). For example, a speed setting of 1.2 implies that the rotational speed of the pump is 20% higher than the normal setting.
Speed Pattern	The time pattern used to control the pump's operation. The multipliers of the pattern are equivalent to speed settings. A multiplier of zero implies that the pump will be shut off during the corresponding time period. Do not check if not applicable. Click the ellipsis button to bring up the Pattern Editor.
Efficiency	Default pump efficiency in percentage. This is a non editable property. To change this property go to Scenario | Analysis Options | Times-Energy.
Efficiency Curve	The curve that represents the pump's wire-to-water efficiency (in percent) as a function of flow rate. This information is used only to compute energy usage. Do not check if not applicable or if the global pump efficiency supplied with the project's Energy Options will be used. Click the ellipsis button to bring up the Curve Editor.
Energy Price	The average or nominal price of energy in monetary units per kw-hr. Used only for computing the cost of energy usage. Leave blank if not applicable or if the global value supplied with the scenario's Energy Options will be used. Monetary units are not explicitly represented.
Price Pattern	The ID label of the time pattern used to describe the variation in energy price throughout the day. Each multiplier in the pattern is applied to the pump's Energy Price to determine a time-of-day pricing for the corresponding period. Leave blank if not applicable or if the global pricing pattern specified in the Scenario's Energy Options will be used. Click the ellipsis button to bring up the Pattern Editor.
OTHER	
Initial Status	State of the pump (open or closed) at the start of the simulation period.
Controls	By pressing this button the application checks if there are control rules associated to the pump. If so, an information window is opened specifying the control rules and highlighting the nodes the pump is dependant upon.

6.6 VALVE PROPERTIES

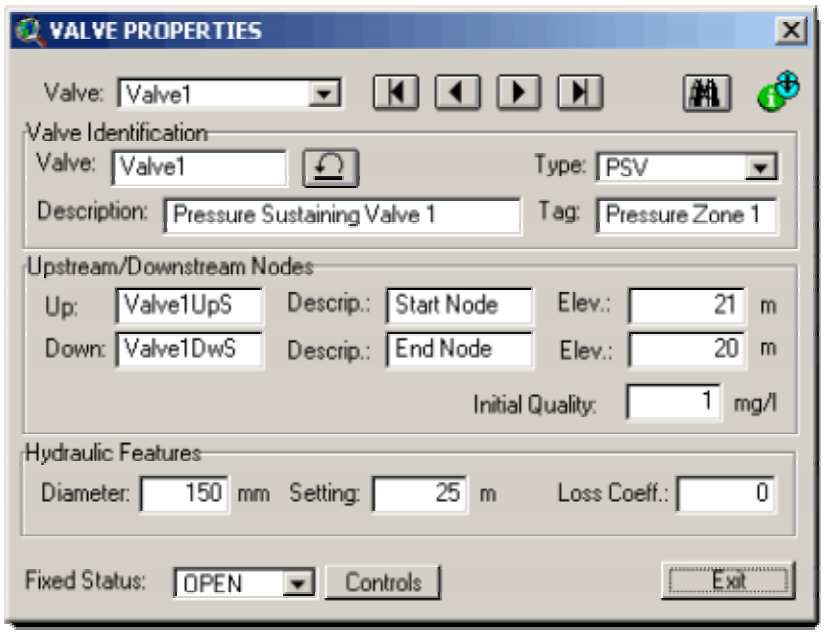


Fig. 51 Valve Properties.

PROPERTY	DESCRIPTION
VALVE IDENTIFICATION	
Valve	A unique label used to identify the valve. It can consist of a combination of up to 15 numerals or characters. It cannot be the same as the ID for any other link. This is a required property.
Flip button	Reverses the direction of the valve.
Description	An optional text string that describes other significant information about the valve.
Type	The valve type (PRV, PSV, PBV, FCV, TCV, or GPV). See the description of the various types of valves below. This is a required property.
Tag	An optional text string used to assign the valve to a category, perhaps based on type or location.
UPSTREAM / DOWNSTREAM NODES	
UPSTREAM	Start node. The ID of the node on the nominal upstream or inflow side of the valve. (PRVs and PSVs maintain flow in only a single direction). This is a non editable property. By default the Upstream Node is named as the Valve ID but adding the 'UpS' suffix (meaning upstream).
Description (Start node)	An optional text string that describes other significant information about the valve upstream node.
Elevation (Start node)	The elevation in feet (meters) above some common reference of the valve start node. This is a required property. Elevation is used only to compute pressure at that point. It does not affect any other computed quantity.

DOWNSTREAM	End Node. The ID of the node on the nominal downstream or discharge side of the valve. This is a non editable property. By default the Downstream Node is named as the Valve ID but adding the 'DwS' suffix (meaning downstream).
Description (End node)	An optional text string that describes other significant information about the valve downstream node.
Elevation (End node)	The elevation in feet (meters) above some common reference of the valve end node. This is a required property. Elevation is used only to compute pressure at that point. It does not affect any other computed quantity.
Initial Quality	Water quality level at the start and end nodes at the start of the simulation period. Can be left blank if no water quality analysis is being made or if the level is zero.
HYDRAULIC FEATURES	
Diameter	The valve diameter in inches (mm). This is a required property.
Setting	A required parameter that describes the valve's operational setting. Valve Type — Setting Parameter PRV — Pressure (psi or m) PSV — Pressure (psi or m) PBV — Pressure (psi or m) FCV — Flow (flow units) TCV — Loss Coefficient (unitless) GPV — ID of head loss curve. Click the ellipsis button to open the curve editor.
Loss Coefficient	Unitless minor loss coefficient that applies when the valve is completely opened. Assumed 0 if left blank.
OTHER	
Fixed Status	Valve status at the start of the simulation. If set to OPEN or CLOSED then the control setting of the valve is ignored and the valve behaves as an open or closed link, respectively. If set to NONE, then the valve will behave as intended. A valve's fixed status and its setting can be made to vary throughout a simulation by the use of control statements. If a valve's status was fixed to OPEN/CLOSED, then it can be made active again using a control that assigns a new numerical setting to it.
Controls	By pressing this button the application checks if there are control rules associated to the valve. If so, an information window is opened specifying the control rules and highlighting the nodes the valve is dependant upon.

6.6.1 Type of Valves

From a hydraulic simulator point of view, valves are links that limit the pressure or flow at a specific point in the network. Their principal input parameters include:

- start and end nodes
- diameter
- setting
- status.

The computed outputs for a valve are flow rate and headloss.

The different types of valves included in GISRed are:

- Pressure Reducing Valve (**PRV**)
- Pressure Sustaining Valve (**PSV**)
- Pressure Breaker Valve (**PBV**)
- Flow Control Valve (**FCV**)
- Throttle Control Valve (**TCV**)
- General Purpose Valve (**GPV**).

- **PRVs** limit the pressure at a point in the pipe network. EPANET computes in which of three different states a PRV can be in:
  - partially opened (i.e., active) to achieve its pressure setting on its downstream side when the upstream pressure is above the setting
  - fully open if the upstream pressure is below the setting
  - closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).

- **PSVs** maintain a set pressure at a specific point in the pipe network. EPANET computes in which of three different states a PSV can be in:
  - partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value
  - fully open if the downstream pressure is above the setting
  - closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).

- **PBVs** force a specified pressure loss to occur across the valve. Flow through the valve can be in either direction. PBV's are not true physical devices but can be used to model situations where a particular pressure drop is known to exist.

- **FCVs** limit the flow to a specified amount. The program produces a warning message if this flow cannot be maintained without having to add additional head at the valve (i.e., the flow cannot be maintained even with the valve fully open).

- **TCVs** simulate a partially closed valve by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which a valve is closed and the resulting head loss coefficient is usually available from the valve manufacturer.

- **GPVs** are used to represent a link where the user supplies a special flow - head loss relationship instead of following one of the standard hydraulic formulas. They can be used to model turbines, well draw-down or reduced-flow backflow prevention valves.

Shutoff (gate) valves and check (non-return) valves, which completely open or close pipes, are not considered as separate valve links but are instead included as a property of the pipe in which they are placed.

Each type of valve has a different type of setting parameter that describes its operating point (pressure for PRVs, PSVs, and PBVs; flow for FCVs; loss coefficient for TCVs, and head loss curve for GPVs).

Valves can have their control status overridden by specifying they be either completely open or completely closed. A valve's status and its setting can be changed during the simulation by using control statements.

Because of the ways in which valves are modeled the following rules apply when adding valves to a network:

- *a PRV, PSV or FCV cannot be directly connected to a reservoir or tank (use a length of pipe to separate the two)*
- *PRVs cannot share the same downstream node or be linked in series*
- *two PSVs cannot share the same upstream node or be linked in series*
- *a PSV cannot be connected to the downstream node of a PRV.*

6.7 EDITING A GROUP OF NETWORK ELEMENTS

To edit a property for a group of network elements:

1. Select the group of elements to be edited using one of the methods described in the section SELECTING ELEMENTS.
2. Select **Edit Model | Group Edit** from the Menu Bar.
3. Define what to edit in the Group Edit dialog form that appears.

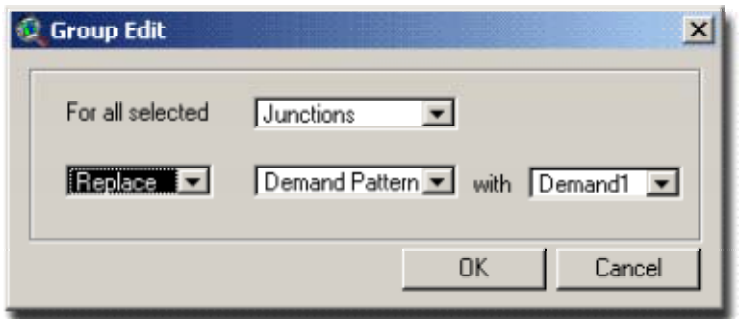


Fig. 52 Group Edit Dialog.

The Group Edit dialog form, shown in Fig. 52, is used to modify a property for a selected group of objects. To use the dialog form:

1. Select a category of element (Junctions, Tanks, Reservoirs, Valves, Pumps or Pipes depending on the selection) to edit.
2. Select the type of change to make - Replace, Multiply, or Add To (depending on the property to change).
3. Select the property to change.
4. Enter the value that should replace, multiply, or be added to the existing value or choose one from the combo box.
5. Click OK to execute the group edit.
6. Once the property has been changed, the application prompts the user with a summary of the edition process.

CHAPTER 7. PATTERNS

A Time Pattern is a collection of multipliers that can be applied to a quantity to allow it to vary over time. Nodal demands, reservoir heads, pump schedules, and water quality source inputs can all have time patterns associated with them. The time interval used in all patterns is a fixed value, set with the Scenario's Time Options (see *Analysis Options*). Within this interval a quantity remains at a constant level, equal to the product of its nominal value and the pattern's multiplier for that time period. Although all time patterns must utilize the same time interval, each can have a different number of periods. When the simulation clock exceeds the number of periods in a pattern, the pattern wraps around to its first period again.

7.1 GISRED PATTERN EDITOR

The **Pattern Editor**, displayed in Fig. 53, edits the properties of a time pattern object. To use the Pattern Editor select one of the five categories (*Demand, Head, Speed, Quality or Price*) of the list and choose an option:

- **New**: Creates a new pattern.
- **Clone**: Clones the current pattern.
- **Del**: Removes the current pattern.
- **Edit**: Edits the current pattern.
- **Save**: Saves the current pattern's data to a file.

➢ To add a **New** pattern:

1. Select '**New**'.
2. A default **pattern ID** is suggested.
3. Choose another **name** alternatively (maximum of 15 numerals or characters).
4. Additionally, you might want to enter a description in the **Description** field.
5. Enter multipliers in the '**Multiplier**' field and press **Enter key**.
6. The multiplier will be added to the current time period (focus).
7. When finished editing, click the **OK** button to accept the pattern or the **Cancel** button to cancel your entries.

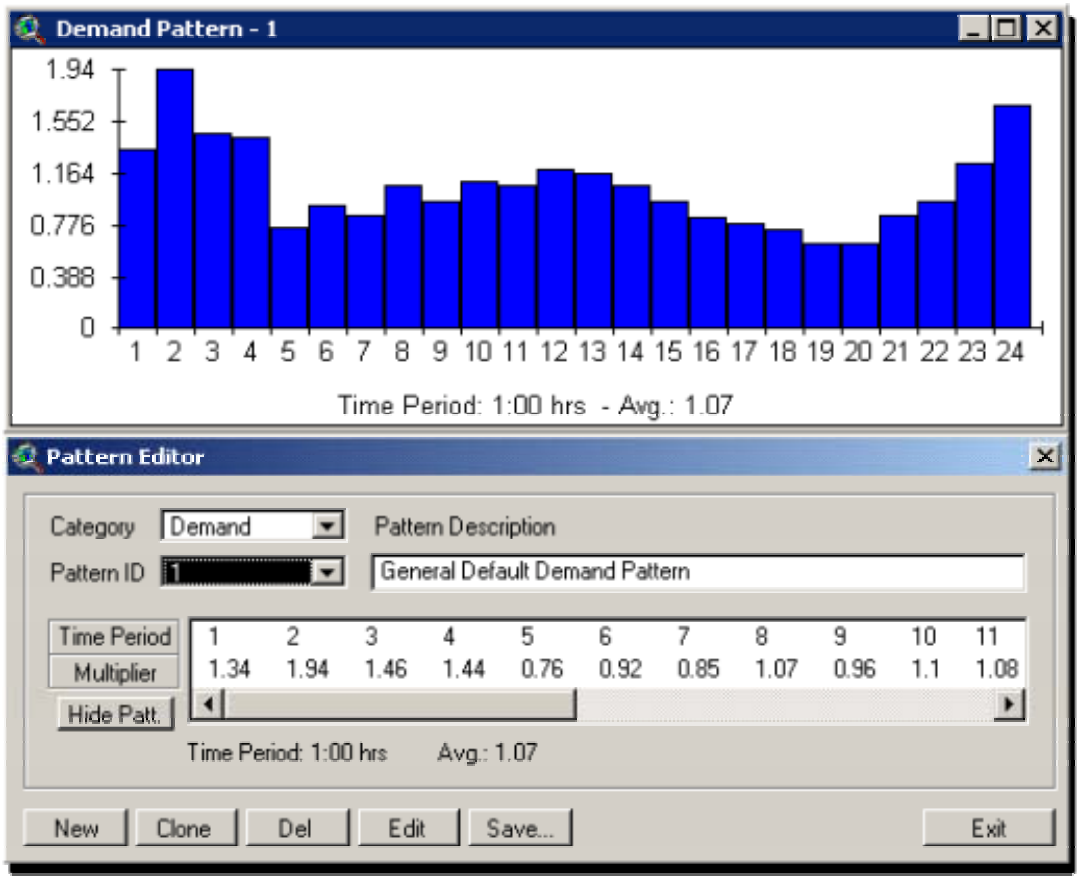
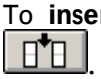


Fig. 53 GISRed Pattern Editor.

NOTES:

- To **insert a new multiplier** in an existing pattern, select a multiplier and press  . A gap will be inserted immediately after the selected multiplier. Enter the new multiplier. Place the cursor in the last time period to go on with the edition.

- To **remove a multiplier**, select the multiplier to be removed and press . The multiplier will be removed and the time periods reordered. Place the cursor in the last time period to go on with the edition.

- The average value is updated as multipliers are entered.

- You can also click the **Load button** to load in pattern data that was previously saved to file. Patterns saved from EPANET can be loaded in GISRed and vice versa.

- To see a chart of the pattern, press the button '**SHOW PATT.**'. Automatically this button turns to '**HIDE PATT.**'. Press again to hide the chart view. If the chart view is opened, as multipliers are entered, the preview chart is redrawn to provide a visual depiction of the pattern.

➢ To **Clone** a pattern:

1. Select the pattern to Clone (Category & Pattern)

2. Press '**Clone**'.
3. An exact copy of the pattern will be created ready to be edited. A default **pattern ID** is suggested.
4. The way to operate at this point is just as described before.
5. When finished editing, click the **OK** button to accept the pattern or the **Cancel** button to cancel your entries.

➢ To **Edit** a pattern:

1. Select the pattern to Edit (Category & Pattern)
2. Press '**Edit**'.
3. The pattern will enter in editing mode.
4. Make the necessary changes.
5. When finished editing, click the **OK** button to accept the pattern or the **Cancel** button to cancel your entries.

➢ To **Delete** a pattern:

1. Select the pattern to delete (Category & Pattern)
2. Press '**Del**'.
3. The pattern will be removed from the list.

➢ To **Save** a pattern:

1. Select the pattern to save (Category & Pattern)
2. Press '**Save…**'.
3. Enter a name for the pattern.
4. The pattern will be saved in a text file with extension '*.pat'.

CHAPTER 8. CURVES

Curves are objects that contain data pairs representing a relationship between two quantities. Two or more objects can share the same curve. An GISRed model can utilize the following types of curves:

- Pump Curve
- Efficiency Curve
- Volume Curve
- Head Loss Curve

PUMP CURVE

A Pump Curve represents the relationship between the head and flow rate that a pump can deliver at its nominal speed setting. Head is the head gain imparted to the water by the pump and is plotted on the vertical (Y) axis of the curve in feet (meters). Flow rate is plotted on the horizontal (X) axis in flow units. A valid pump curve must have decreasing head with increasing flow.

GISRed, as in EPANET, will use a different shape of pump curve depending on the number of points supplied:

- **Single-Point Curve** - A single-point pump curve is defined by a single head-flow combination that represents a pump's desired operating point. GISRED adds two more points to the curve by assuming a shutoff head at zero flow equal to 133% of the design head and a maximum flow at zero head equal to twice the design flow. It then treats the curve as a three-point curve.

- **Three-Point Curve** - A three-point pump curve is defined by three operating points: a Low Flow point (flow and head at low or zero flow condition), a Design Flow point (flow and head at desired operating point), and a Maximum Flow point (flow and head at maximum flow). GISRED tries to fit a continuous function of the form $h_G = A - Bq^C$ through the three points to define the entire pump curve. In this function, h_G = head gain, q = flow rate, and A, B, and C are constants.

- **Multi-Point Curve** – A multi-point pump curve is defined by providing either a pair of head-flow points or four or more such points. GISRED creates a complete curve by connecting the points with straight-line segments.

For variable speed pumps, the pump curve shifts as the speed changes. The relationships between flow (*Q*) and head (*H*) at speeds N_1 and N_2 are:

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2} \qquad \frac{H_1}{H_2} = \left(\frac{N_1}{N_2}\right)^2$$

EFFICIENCY CURVE

An Efficiency Curve determines pump efficiency (*Y in percent*) as a function of pump flow rate (*X in flow units*). Efficiency should represent wire-to-water efficiency that takes into account mechanical losses in the pump itself as well as electrical losses in the pump's motor. The curve is used only for energy calculations. If not supplied for a specific pump then a fixed global pump efficiency will be used.

VOLUME CURVE

A Volume Curve determines how storage tank volume (*Y in cubic feet or cubic meters*) varies as a function of water level (*X in feet or meters*). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. The lower and upper water levels supplied for the curve must contain the lower and upper levels between which the tank operates. An example of a tank volume curve is given below.

HEADLOSS CURVE

A Headloss Curve is used to describe the headloss (*Y in feet or meters*) through a General Purpose Valve (**GPV**) as a function of flow rate (*X in flow units*). It provides the capability to model devices and situations with unique headloss-flow relationships, such as reduced flow - backflow prevention valves, turbines, and well draw-down behaviour.

8.1 GISRED CURVE EDITOR

The **Curve Editor**, displayed in Fig. 54, edits the properties of a curve object. To use the Curve Editor select one of the four curve types (*Pump, Efficiency, Volume, Headloss*) of the list and choose an option:

- **New**: Creates a new curve.
- **Clone**: Clones the current curve.
- **Del**: Removes the current curve.
- **Edit**: Edits the current curve.
- **Save**: Saves the current curve's data to a file.

➢ To add a new curve:

1. Select '**New**'.
2. A default **Curve ID** is suggested.
3. Choose another **name** alternatively (maximum of 15 numerals or characters).
4. Additionally, you might want to enter a description in the **Description** field.
5. Enter the X-Y values and press **Add Coordinates**.
6. The pair of values will be added to the current coordinate list cell (focus).
7. When finished editing, click the **OK** button to accept the curve or the **Cancel** button to cancel your entries.

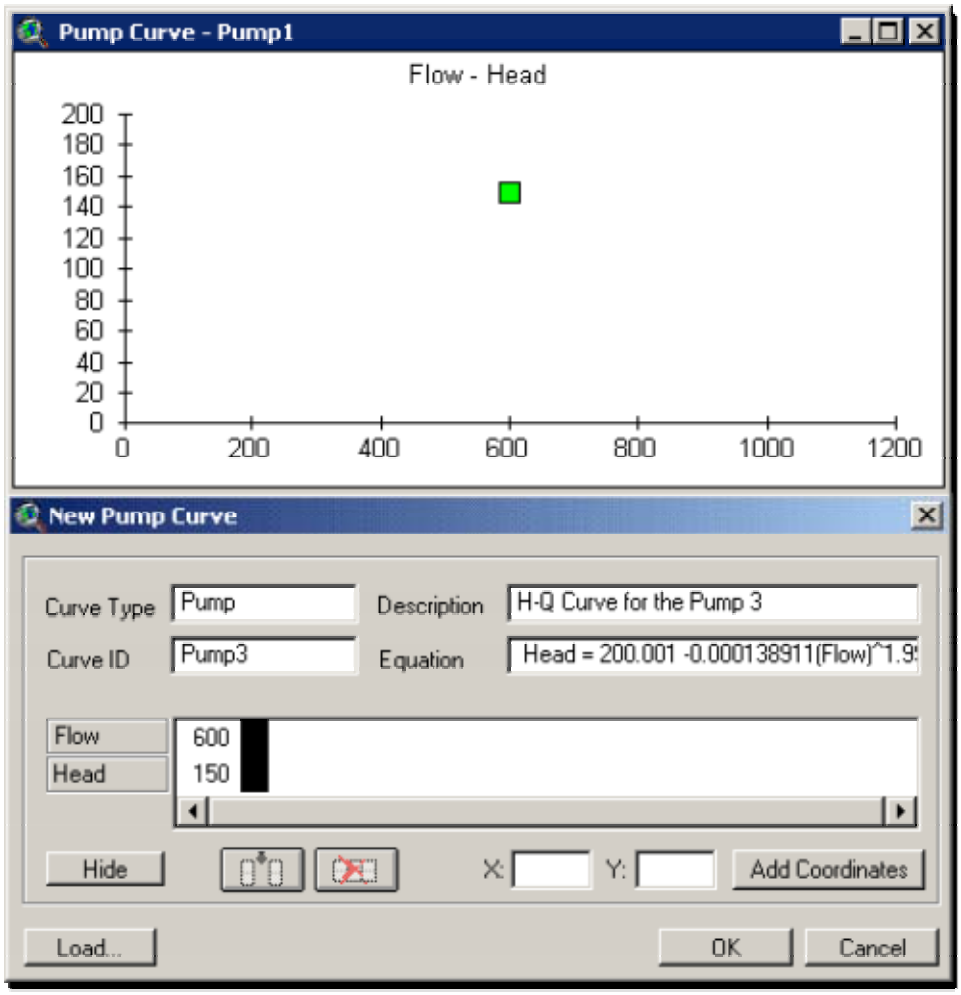


Fig. 54 GISRed Curve Editor.

NOTES:

- To **insert a new point** in an existing curve, select a cell and press . A gap will be inserted immediately after the selected coordinate. Enter the new pair of values and press **Add Coordinates**. Place the focus after the last X-Y value to go on with the edition.

- To **remove an existing point**, select the cell to be removed and press . The X-Y value will be removed. Place the focus after the last X-Y value to go on with the edition.

- To see a chart of the points, press the button '**SHOW**'. Automatically this button turns to '**HIDE**'. Press again to hide the chart view. If the chart view is opened, as you enter new coordinates in the X-Y list box the points are redrawn in the preview window.

- For single and three-point pump curves, the equation generated for the curve will be displayed in the Equation box.

- You can also click the **Load button** to load in curve data that was previously saved to file. Curves saved from EPANET can be loaded in GISRed and vice versa.

➢ To **Clone** a curve:

1. Select the curve to Clone (Type & Curve)
2. Press '**Clone**'.
3. An exact copy of the curve will be created ready to be edited. A default **curve ID** is suggested.
4. The way to operate at this point is just as described before.
5. When finished editing, click the **OK** button to accept the curve or the **Cancel** button to cancel your entries.

➢ To **Edit** a curve:

1. Select the curve to Edit (Type & Curve)
2. Press '**Edit**'.
3. The curve will enter in editing mode.
4. Make the necessary changes.
5. When finished editing, click the **OK** button to accept the curve or the **Cancel** button to cancel your entries.

➢ To **Delete** a curve:

1. Select the curve to delete (Type & Curve)
2. Press '**Del**'.
3. The curve will be removed from the list.

➢ To **Save** a curve:

5. Select the curve to save (Type & Curve)
6. Press '**Save…**'.
7. Enter a name for the curve.
8. The curve will be saved in a text file with extension '*.crv'.

CHAPTER 9. CONTROLS

Controls are statements that determine how the network is operated over time. They specify the status of selected links as a function of time, tank water levels, and pressures at selected points within the network. There are two categories of controls that can be used:

- Simple Controls
- Rule-Based Controls

To define controls in GISRed:

1. Select **Edit Model | Controls.**
2. The **Control Rules** dialog appears.
3. Choose the category of controls you want to define.
4. Press **OK**.

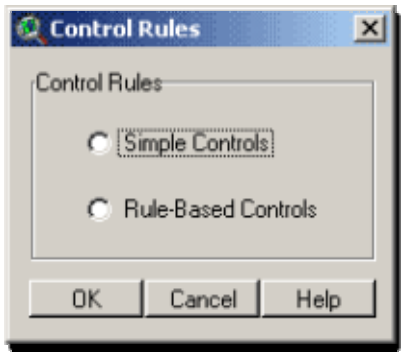


Fig. 55 Control Rules Dialog.

9.1 SIMPLE CONTROLS

Simple controls change the status or setting of a link based on:

- the water level in a tank,
- the pressure at a junction,
- the time into the simulation,
- the time of day.

They are statements expressed in one of the following three formats:

```
LINK linkID status IF NODE nodeID ABOVE/BELOW value
LINK linkID status AT TIME time
LINK linkID status AT CLOCKTIME clocktime AM/PM
```

where:

linkID	=	*a link ID label*
status	=	*OPEN or CLOSED, a pump speed setting, or a control valve setting*
nodeID	=	*a node ID label*
value	=	*a pressure for a junction or a water level for a tank*
time	=	*a time since the start of the simulation in decimal hours or in hours:minutes format*

clocktime = *a 24-hour clock time (hours:minutes)*

Some examples of simple controls are:

CONTROL STATEMENT	MEANING
`LINK 12 CLOSED IF NODE 23 ABOVE 20`	(Close Link 12 when the level in Tank 23 exceeds 20 ft.)
`LINK 12 OPEN IF NODE 130 BELOW 30`	(Open Link 12 if the pressure at Node 130 drops below 30 psi)
`LINK 12 1.5 AT TIME 16`	(Set the relative speed of pump 12 to 1.5 at 16 hours into the simulation)
`LINK 12 CLOSED AT CLOCKTIME 10 AM` `LINK 12 OPEN AT CLOCKTIME 8 PM`	(Link 12 is repeatedly closed at 10 AM and opened at 8 PM throughout the simulation)

There is no limit on the number of simple control statements that can be used.

Note: *Level controls are stated in terms of the height of water above the tank bottom, not the elevation (total head) of the water surface.*

Note: *Using a pair of pressure controls to open and close a link can cause the system to become unstable if the pressure settings are too close to one another. In this case using a pair of Rule-Based controls might provide more stability.*

9.1.1 GISRed Simple Controls EDITOR

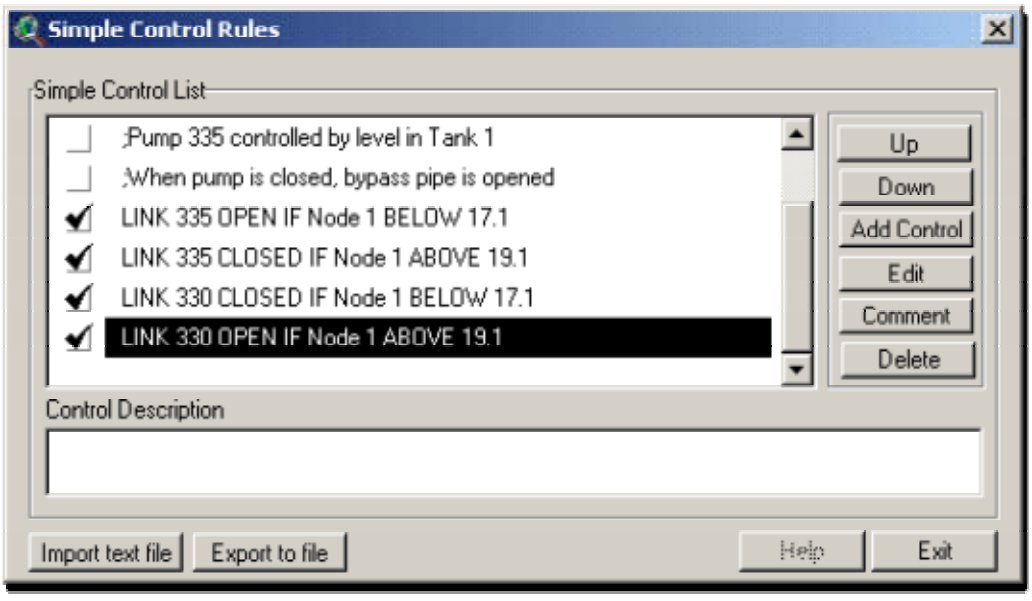


Fig. 56 GISRed Simple Controls Editor.

➢ To **add a new simple control** to the list:

1. Click on the **Add Control** button.

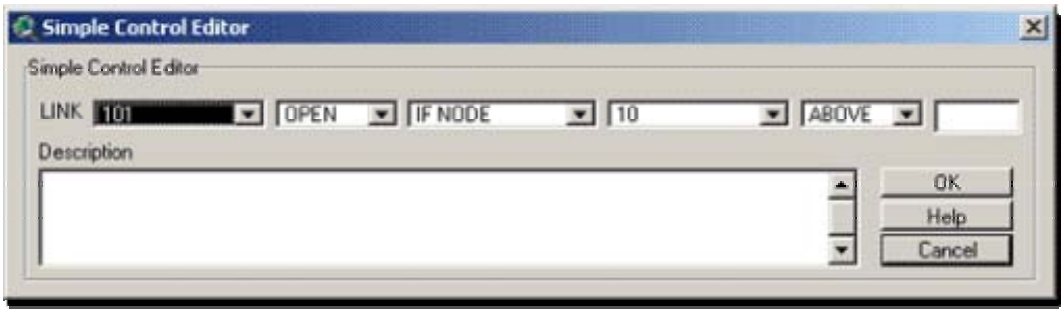
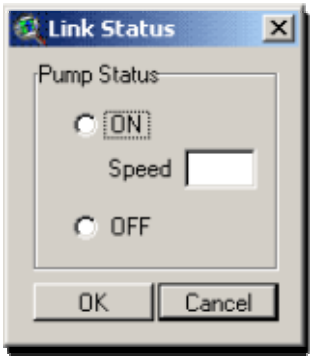
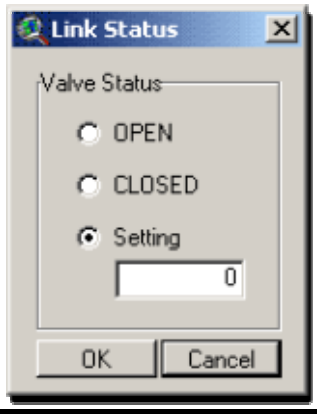


Fig. 57 Adding a new simple control.

2. Choose the **Link ID** from the list.
3. Enter the **Link Status**. Depending on the type of link:

TYPE OF LINK	STATUS
9.1.1.1 PIPE	OPEN CLOSED
PUMP	PUMP SPEED SETTING 
VALVE	9.1.1.2 VALVE SETTING 

4. Choose the **condition** to apply (**`IF NODE`**, **`AT TIME`** or **`AT CLOCKTIME`**) Depending upon the condition the dialog will change accordingly.
5. Enter the **Node ID**, and a **pressure** for a junction **or** a **water level** for a tank if '**`IF NODE`**' was chosen.
6. Enter a **time** since the start of the simulation in *decimal hours* or in *hours:minutes* format if '**`AT TIME`**' was the choice.
7. Enter a **24-hour clock time** (*hours:minutes*) if '**`AT CLOCKTIME`**' was selected.
8. Additionally, enter a **description** for the simple control.
9. Press **OK**.

The simple control will be added/inserted just below the currently focused control/comment.

- To **edit** an existing simple control:
  1. **Select** an existing simple control from the list.
  2. Press the **Edit** button.
  3. Change the fields as necessary.
  4. Click **OK**.

  The control will be updated consequently.

- To **move up and down** an existing simple control:
  1. **Select** an existing simple control from the list.
  2. Press the **UP and DOWN** buttons to change the position of the control in the list.

  The control will be placed consequently.

- To add a **comment**:
  1. Press the **Comment** button.
  2. Enter a comment
  3. Click **OK**.

  The comment will be added/inserted just below the currently focused control/comment.

- To **activate/deactivate** a simple control:
  1. Toggle on (☑) the checkmark on the left of the simple control to activate the control. The Control **will be taken into account** when simulating the network.
  2. Toggle off (☐) the checkmark on the left of the simple control to deactivate the control. The Control **will not be taken into account** when simulating the network.

- To **delete** a control or a comment (one at a time):
  1. Select one **Control/Comment**.
  2. Press the '**Delete**' button

  The control/comment will be removed from the list.

Importing/Exporting Simple Controls

- To **Import Simple Controls** from a text file:

  1. Open the **Simple Controls Editor**.
  2. Press on **Import Text File**.
  3. Choose the text file that contains the Simple Controls.
  4. Accept.

  The Simple Controls will be added to the database and displayed in the editor window. At the end of the process a summary is displayed. The summary specifies how many rules were imported and how many were not. A list of the non-imported controls will be given.

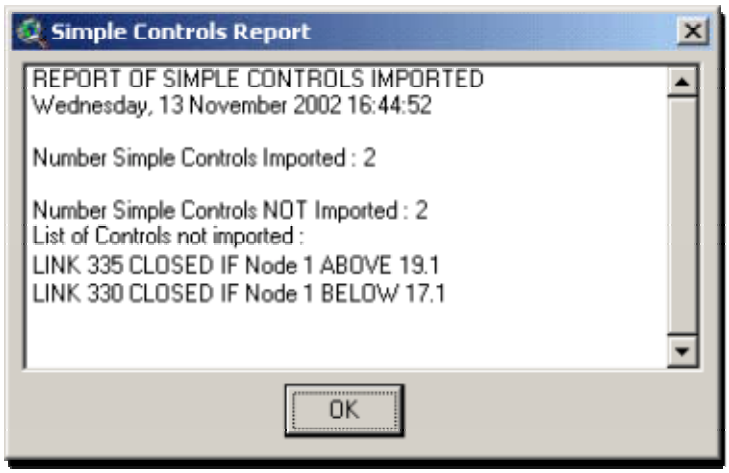

Simple Controls Report
REPORT OF SIMPLE CONTROLS IMPORTED
Wednesday, 13 November 2002 16:44:52
Number Simple Controls Imported : 2
Number Simple Controls NOT Imported : 2
List of Controls not imported :
LINK 335 CLOSED IF Node 1 ABOVE 19.1
LINK 330 CLOSED IF Node 1 BELOW 17.1
OK

Fig. 58 Report of Simple Controls Imported.

NOTE: Any text file can be added to the editor. So, care should be taken that the format of the imported text is accepted as rule-based controls (see the format in previous sections).

➢ To **Export the Simple Controls** list to a text file:

1. Open the **Simple Controls Editor**.
2. Press **Export to File**.
3. Enter a name for the text file to be exported.
4. Accept.

The **Simple Controls** will be saved to a file with *.scn extension.

9.2 RULE-BASED CONTROLS

Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network after an initial hydraulic state of the system is computed.

Each rule is a series of statements of the form:

```
RULE ruleID
IF  condition_1
AND condition_2
OR  condition_3
AND condition_4
etc.
THEN action_1
AND  action_2
etc.
ELSE action_3
AND  action_4
etc.
PRIORITY value
```

where:

RuleID	=	an ID label assigned to the rule
conditon_n	=	a condition clause
Action_n	=	an action clause
Priority	=	a priority value (e.g., a number from 1 to 5)

☞ **Condition Clause Format:**

A condition clause in a Rule-Based Control takes the form of:

object id attribute relation value

where

object	=	a category of network object
Id	=	the object's ID label
Attribute	=	an attribute or property of the object
relation	=	a relational operator
value	=	an attribute value

Some example conditional clauses are:

```
JUNCTION 23 PRESSURE > 20
TANK T200 FILLTIME BELOW 3.5
```

```
LINK 44 STATUS IS OPEN
SYSTEM DEMAND >= 1500
SYSTEM CLOCKTIME = 7:30 AM
```

The Object keyword can be any of the following:

NODE	**LINK**	**SYSTEM**
JUNCTION	**PIPE**	
RESERVOIR	**PUMP**	
TANK	**VALVE**	

When **SYSTEM** is used in a condition no ID is supplied.

The following attributes can be used with **Node-type elements**:

DEMAND

HEAD

PRESSURE

The following attributes can be used with **Tanks**:

LEVEL

FILLTIME (hours needed to fill a tank)

DRAINTIME (hours needed to empty a tank)

These attributes can be used with **Link-Type elements**:

FLOW

STATUS (**OPEN**, **CLOSED**, or **ACTIVE**)

SETTING (pump speed or valve setting)

The **SYSTEM** object can use the following attributes:

DEMAND (total system demand)

TIME (hours from the start of the simulation expressed either as a decimal number or in hours:minutes format)

CLOCKTIME (24-hour clock time with **AM** or **PM** appended)

Relation operators consist of the following:

=	**IS**
<>	**NOT**
<	**BELOW**
>	**ABOVE**
<=	>=

☞ **Action Clause Format:**

An action clause in a Rule-Based Control takes the form of:

object id STATUS/SETTING IS value

where

object	=	**LINK**, **PIPE**, **PUMP**, or **VALVE** keyword
id	=	the object's ID label
value	=	a status condition (**OPEN** or **CLOSED**), pump speed setting, or valve setting

Some example action clauses are:

```
LINK 23 STATUS IS CLOSED
PUMP P100 SETTING IS 1.5
VALVE 123 SETTING IS 90
```

NOTES:

a. Only the **RULE**, **IF** and **THEN** portions of a rule are required; the other portions are optional.

b. When mixing **AND** and **OR** clauses, the **OR** operator has higher precedence than **AND**, i.e.,
```
IF A or B and C
```
is equivalent to
```
IF (A or B) and C.
```
If the interpretation was meant to be
```
IF A or (B and C)
```
then this can be expressed using two rules as in
```
IF A THEN ...
IF B and C THEN ...
```

c. The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

EXAMPLES:

Example 1:

This set of rules shuts down a pump and opens a by-pass pipe when the level in a tank exceeds a certain value and does the opposite when the level is below another value.

```
RULE 1
IF   TANK   1 LEVEL ABOVE 19.1
THEN PUMP 335 STATUS IS CLOSED
AND  PIPE 330 STATUS IS OPEN

RULE 2
IF   TANK   1 LEVEL BELOW 17.1
THEN PUMP 335 STATUS IS OPEN
AND  PIPE 330 STATUS IS CLOSED
```

Example 2:

These rules change the tank level at which a pump turns on depending on the time of day.

```
RULE 3
IF   SYSTEM CLOCKTIME >= 8 AM
AND  SYSTEM CLOCKTIME < 6 PM
AND  TANK 1 LEVEL BELOW 12
THEN PUMP 335 STATUS IS OPEN

RULE 4
IF   SYSTEM CLOCKTIME >= 6 PM
OR   SYSTEM CLOCKTIME < 8 AM
AND  TANK 1 LEVEL BELOW 14
THEN PUMP 335 STATUS IS OPEN
```

9.2.1 GISRed Rule-Based Controls EDITOR

The **Rule-Based Control Editor**, shown in Fig. 59, is a text editor window that presents an interface for displaying and/or editing multiple lines of rule-based control text. The text editor window contains buttons to import controls from a text file and to export the rules to a text file.

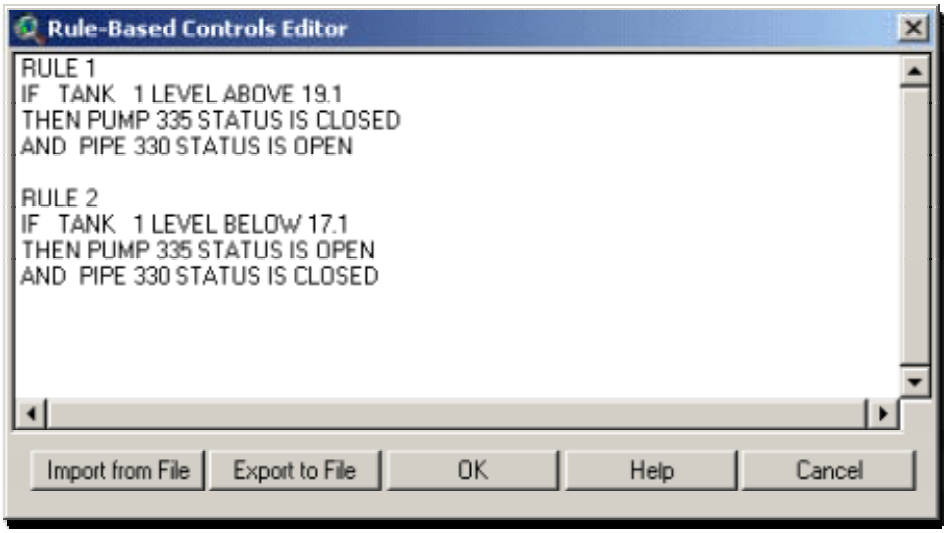


Fig. 59 Rule-based Controls Editor.

➢ To **Import** Rules from a text file:

5. Open the **Rule-Based Controls Editor**.
6. Press **Import from File**.
7. Choose the text file that contains the Control Rules.
8. Accept.

The Rules will be added to the editor window.

NOTE: Any text file can be added to the editor. So, care should be taken that the format of the imported text is accepted as rule-based controls (see the format in previous sections).

➢ To **Export** Rules to a text file:

5. Open the **Rule-Based Controls Editor**.
6. Press **Export to File**.
7. Enter a name for the text file to be exported.
8. Accept.

The **Rule-Based Controls** will be saved to a file with *.rbc extension.

CHAPTER 10. GISRED TOC THEME MANAGER

10.1 TOC (TABLE OF CONTENTS)

As any view document, each scenario has its own Table of Contents that lists the themes in the scenario. You can change how the table of contents looks by choosing TOC Style from the View menu. You also use the Table of Contents to control how the view is drawn:

The Table of Contents shows:

☞ **The name of each theme in the view**
Themes can be given any name. By default, a theme is named after the data source it represents, such as "Streets" or "Parcels". You can give themes longer, more descriptive names. To change the theme's name, select **Theme | Properties** and enter a new name in the *Theme Name* field.

☞ **The legend for each theme**
A theme's legend shows the symbols and colours used to draw the theme. A theme may be drawn using one symbol, or a range of different symbols and colours may be used in order to classify the features in the theme. There are three ways to edit the theme's legend: 1) selecting **Theme | Edit Legend**, 2) Clicking or 3) Double-clicking on the theme's legend itself.

☞ **Whether a theme is on or off**
Each theme has a check box to its left that indicates whether the theme is currently drawn in the scenario/view. You control which themes are drawn in your scenario/view by simply checking these boxes.

☞ **The order the themes are drawn in**
The theme at the top of the Table of Contents is drawn on top of those below it. Themes that form the background to your view are therefore at the bottom of the list. Simply drag themes up and down in the Table of Contents to change the order they are drawn in.

☞ **Which themes are active**
When you make themes active you choose which themes you wish to work with. When a theme is active it is highlighted in the Table of Contents. Simply click a theme's name or legend to make it active. To make more than one theme active, hold down *SHIFT* when you click on the themes.

Most of the operations you can perform on a scenario/view work on the active theme(s). For example, when you select features on a scenario/view, features are only selected from the active theme(s).

☞ **Which theme is editable**
A dashed line around the theme's check box indicates that you are currently editing the features in the theme. Only themes based on a shapefile can be edited.

☞ **Hiding the Table of Contents**
To hide the Table of Contents, drag its right border all the way to the left. The scenario/view will redraw to fill the whole window. To show the Table of Contents again, drag its border back to the right again.

☞ **Hiding a theme's legend**
The Table of Contents normally shows the legend of each theme in the scenario/view. However, you can save space in the Table of Contents by hiding legends. This is especially useful when a scenario/view contains many themes. To hide a theme's legend, make the theme active and then choose ***Hide/Show Legend*** from the ***Theme*** menu. When a legend is hidden, the name of the theme and its check box remain visible.

☞ **Cutting, copying, and pasting themes**
You can copy and paste themes back into the same scenario/view or into another scenario/view. To remove a theme from a scenario/view, simply cut it from the Table of Contents. The scenario/view will automatically redraw without the theme you cut.

Choose TOC Style from the View menu to change display settings such as line flatness, symbol length and text symbol for the TOCs in all scenarios/views in your project.

10.2 TOC THEME MANAGER

On the other hand, GISRed offers a special dialog to manage the themes which are displayed in a scenario. This manager allows the user to show/hide themes in the TOC preserving the current legend. In this way, the user can add themes to the TOC (general ArcView themes or GISRed custom-built themes), modify their legends and remove other themes without losing the legend associated to them as well as other relevant information attached to the theme. To make it possible, GISRed generates an ODB file stored in the database which contains all the details of all themes in the scenario.

There are three ways to **open** the **TOC Theme Manager**:

1. Selecting **Scenario | TOC Manager** from the menu bar.
2. Pressing in the GISRed tool bar.
3. Using the **popup menu** on the scenario. Right-clicking and selecting TOC Manager.

The theme manager is shown in the next figure.

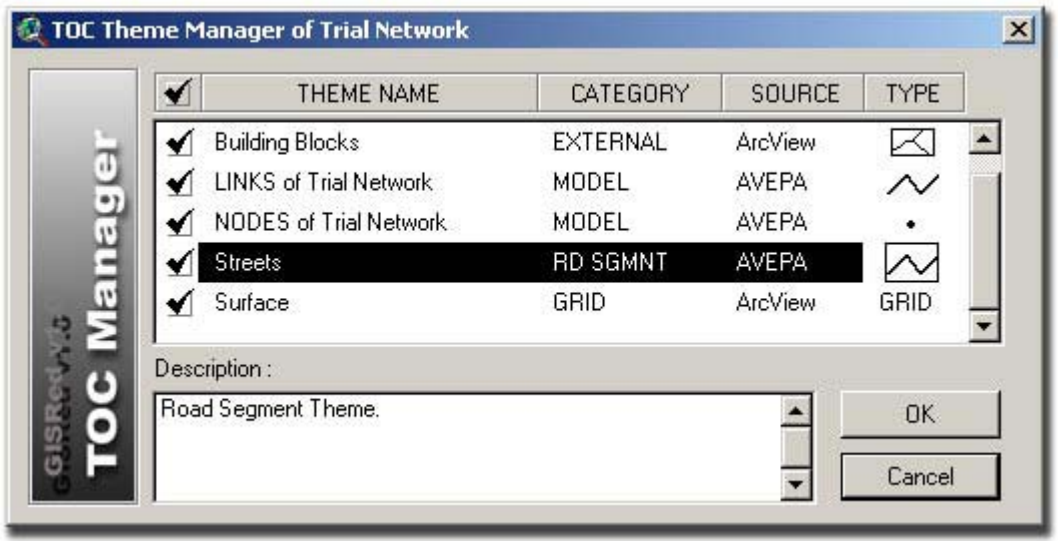


Fig. 60 GISRed TOC Theme Manager.

The TOC Theme Manager shows a list of the themes which are currently on the scenario table of contents (those themes toggled on) and also those themes that were added to the scenario and removed from it at some later time (those themes toggled off). A series of information is given for each theme:

- **ON(☑)/OFF(☐)**: The mark indicates that the theme is visible in the TOC. If the mark is off, the theme exists on disk but is not on the TOC.

- **Theme Name**: Name of the theme.

- **Category**: Type of Theme. There are several types of themes. The EXTERNAL category indicates that the theme is a general shapefile with a structure not controlled by GISRed. Those custom-built by GISRed are:
  - MODEL (basic network model themes, links and nodes),
  - RD SGMNT (road segment themes, used to geocode streets)
  - STREETS (street themes, used to build street themes without the need to geocode)
  - GRID (grid themes, surfaces, etc.)
  - AUX (auxiliary themes. Used normally to add information, they are not modelling-oriented)
  - IMG CATALOG (image catalog themes).

- **Source**: This field indicates whether the theme is controlled by GISRed or not. If it is custom-built from the application, the value should be '*GISRED'*. Otherwise, '*ArcView*' is the default value.

- **Type**: This field specifies with an icon, whether the theme is a polyline theme, a point theme, a grid or a polygon theme.

- **Description**: It presents a description for the selected theme. This field is editable.

➢ To **Add/Remove** a theme to/from the scenario table of contents:

1. **Toggle on/off** the check box on the left.
2. Click **OK**.
3. The theme is added/removed to/from the TOC.

When a theme is removed this way, it is not removed from disk, it is just removed from the scenario TOC and will continue appearing on the TOC theme manager for its future use.

10.3 GISRED CUSTOM-BUILT THEMES

An ArcView theme is a set of geographic features in a view that represents a source of geographic data such as: a spatial data source (an ARC/INFO coverage, ArcView shapefile), a CAD drawing, an image, etc. This data can be on a local disk or accessed across a network. A theme points to the geographic data it represents but does not contain the data itself. Normally, a theme represents all the features in a particular feature class, but it is also possible to define a feature selection property for a theme so that it only represents a specific subset of these features. Themes have a number of other properties that can be set to control their characteristics; for example, the range of scales at which the theme will be drawn on the view. Each theme has its own legend displayed in the Table of Contents and a theme's legend controls how the theme is displayed on the view.

In addition to the link and node themes, an GISRed scenario supports any sort of standard ArcView theme. Particularly, it is possible to create some 'under control' themes such as,

device and pipe themes which include mainly extra hydraulic information without any topology. These themes are considered GIS themes because of their particular tabular structure and are useful to transfer data from and to the model. Other special themes are for instance, a catalog of images that can be used as a backdrop for the network, and a street address theme of the urban area, with geocoding capabilities.

- To **create an GISRed custom-built theme**:

1. Select **Scenario | Add Customized Theme** from the menu bar, or alternatively press from the GISRed tool bar.
2. The dialog of the next figure is opened.
3. Choose the type of theme you want to create from the category list.
4. Enter a name for the theme.
5. Enter a description (optional).
6. Press OK.

The new theme will be created in the database and added to the TOC.

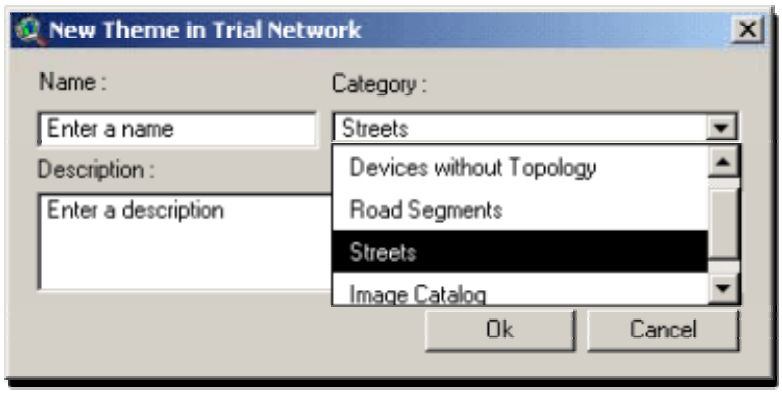


Fig. 61 Adding a new GISRed Theme.

List of customized themes that can be added to the scenario:

- **PIPES WITHOUT TOPOLOGY**: Pipe theme (polyline type) used to add pipes which are not linked to the network model. This theme does not manage any topology. The attribute table associated to the theme contains fields such as sector, zone, manufacturer, material, nominal diameter, internal diameter, external diameter, installation date, owner, street it belongs to, etc.
- **DEVICES WITHOUT TOPOLOGY**: Device theme (point type) used to represent elements such as hydrants, shut-off valves, etc. which are not linked to the network model. This theme does not manage any topology. The attribute table associated to the theme contains fields such as sector, zone, manufacturer, angle, etc.
- **ROAD SEGMENTS**: Special polyline theme that contains the necessary matchable address attributes for geocoding. It can be used to generate routes and perform network analysis as well as build street segments for locating postal addresses.
- **STREETS**: It is a polyline theme in which the streets are represented by a unique polyline not by a sum of segments.
- **IMAGE CATALOG**: An image catalog is an organized collection of spatially referenced geographic images that can be accessed as one logical image theme. Image catalogs typically contain images that depict the same thematic information for a given geographic area of interest. See the section '*Creating an Image Catalog in GISRed*' to learn more about how to work with GISRed image catalogs.

CHAPTER 11. QUERYING THE NETWORK

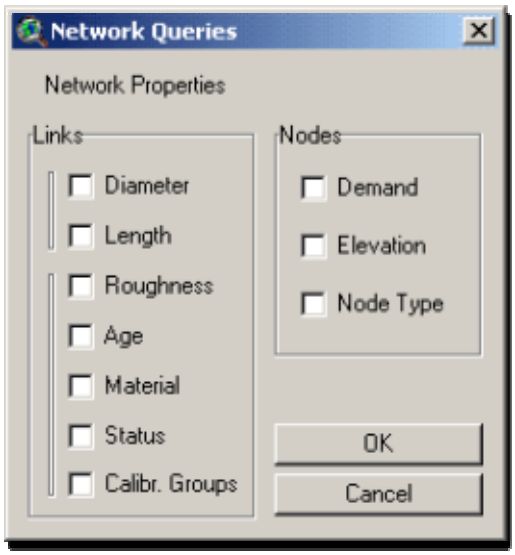


Fig. 62 Network Queries Dialog.

A network query adds a new theme of nodes or links on the network scenario classified by a specific feature (e.g., nodes classified by base demand, links classified by diameter). To submit a network query:

1. Select **Model Tools | Queries** from the menu bar or alternatively, press from the GISRed tool bar.
2. Check-mark those features you want to use to classify the corresponding theme.
3. Press **OK**. A theme for each one of the options selected will be added to the TOC.
4. Press **Cancel** to close the dialog.

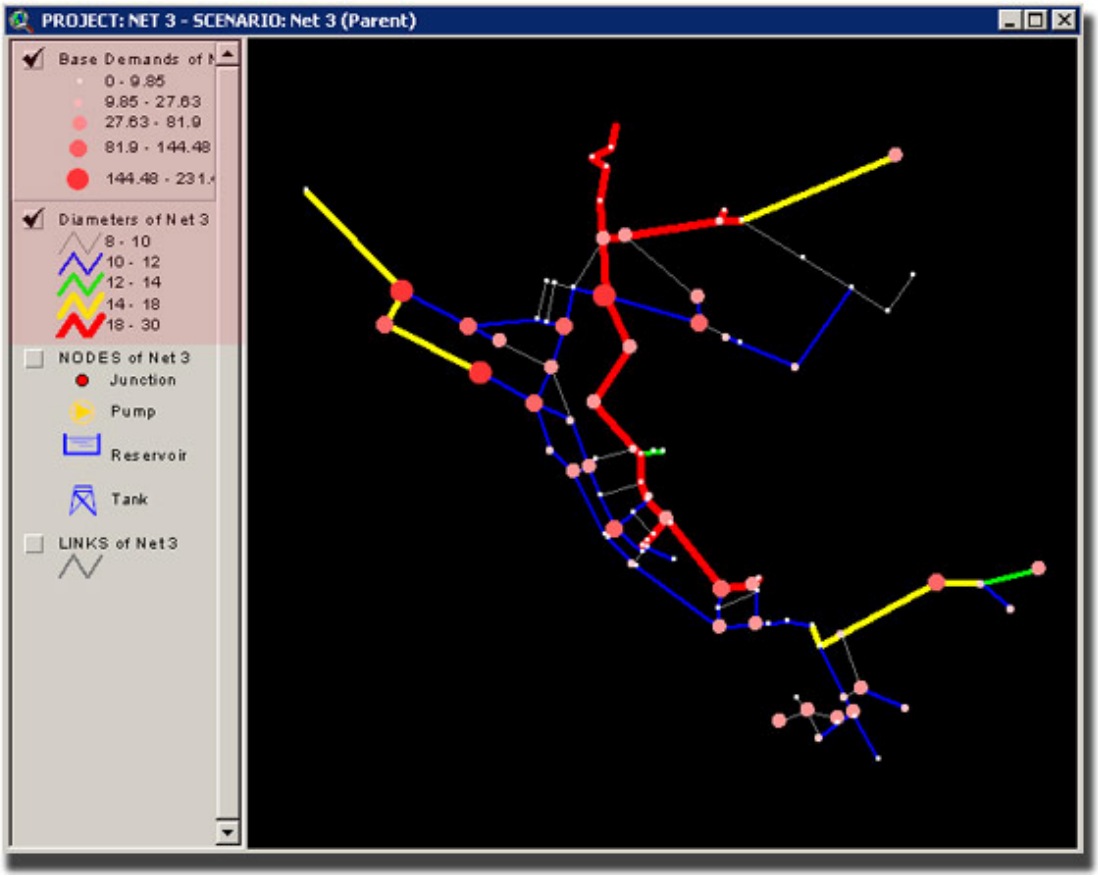


Fig. 63 Network Queries. Diameters and Base Demands.

CHAPTER 12. NETWORK TOOLS

12.1 CHECKING TOOLS

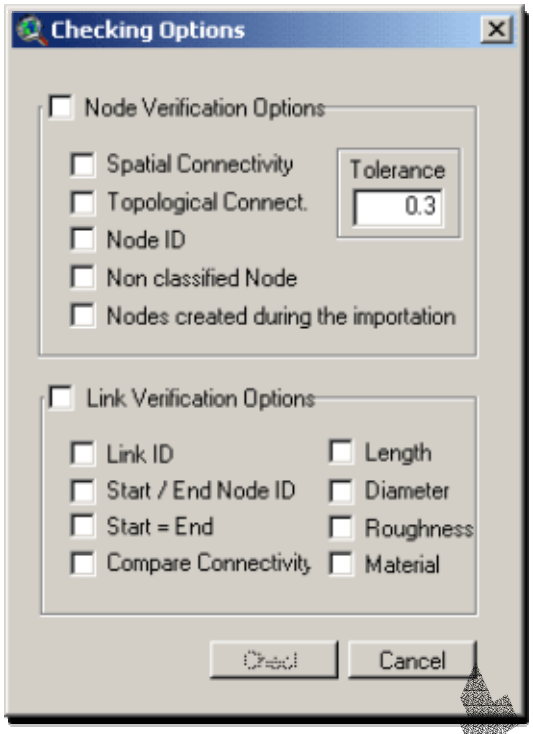


Fig. 64 Network Checking Options.

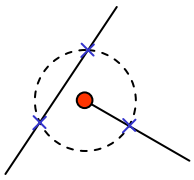
GISRed offers additional tools to check layout inconsistencies or errors on network features such as lack of element identifiers, pipe length, diameter, roughness, material, etc. After the verification process, if some node or link errors were detected, a graphical error theme will be added to the scenario's table of contents classified by the actual errors.

To check the network:

1. Select **Edit Model | Data Verification**.
2. Choose **Node** or /and **Link Verification Options**.
3. **Check-mark** those options you want to verify.
4. Press the '**Check**' button.

Still Not Available

12.1.1 Verification Options

☑ NODE VERIFICATION OPTIONS	
Spatial Connectivity	Checks those points at which there might be spatial inconsistencies within a certain tolerance. This option is usually used after an importation process. For instance, imagine that after a CAD importation there is something like this:  After using this option, an error node would appear at the node of the picture, because the tool detects 2 very close pipes and the connectivity degree at that node is only one.
Topological Connectivity	Checks those points at which there might be topological inconsistencies. This option is usually used after an importation process. The process is exactly the same as the previous one, but it checks the topology of the nodes and pipes involved (*FROM* node & *TO* node), if the topology is correct, it goes on with the process.
Node ID	Checks if there is some node ID missing.
Non Classified Node	*Not available* (*used in a previous version*).

Nodes created during the importation	During the *shapefile importation process* every time a line is imported, the end nodes are created automatically by the application to preserve the topology (distinguishable by the attribute NodetypeID = 99). There is an option in the shapefile importation dialog to compare the nodes that were created automatically against another node theme. Those nodes that did not match within a tolerance after the comparison, can be detected by this option.
☑ LINK VERIFICATION OPTIONS	
Link ID	Checks if there is any pipe ID missing.
Start/End Node ID	Checks if either the start or the end node ID of a pipe are missing.
Start = End	Checks if a pipe starts and ends at the same point (same starting and ending node).
Compare Connectivity	
Length	Checks if there are pipes with no length defined.
Diameter	Checks if there are pipes with no diameter defined.
Roughness	Checks if there are pipes with no roughness defined.
Material	Checks if there are pipes with no material defined.

12.2 CONNECTIVITY TOOLS

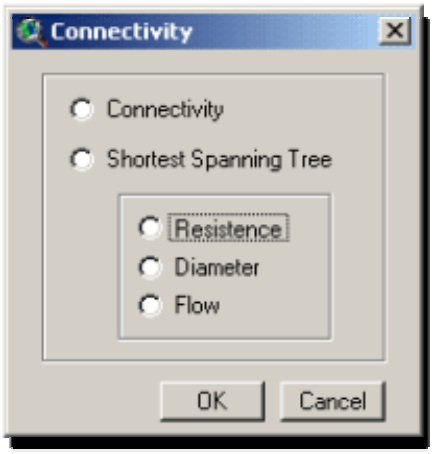


Fig. 65 Connectivity Dialog.

Another useful tool included in GISRed Extension is the Connectivity Checker. This tool enables the user to perform connectivity tests based upon graph theory algorithms (such as Kruskal's algorithm).

It offers several possibilities:

- Find out disconnections in the network.
- Find out one of the possible network trees.
- Find out the shortest spanning tree based on the hydraulic resistance of the pipes, the diameters, or the flow rate.

➢ To check the **connectivity** of the network and find out the number of **unconnected networks** (sub-networks):

1. Select **Model Tools | Connectivity**.
2. Select **Connectivity** from the emerging dialog.
3. Press **OK**.
4. A new theme will be created and added to the scenario classified by the number of subgraphs (sub-networks).

The next figure shows an example in which there are 7 disconnected sub-networks.

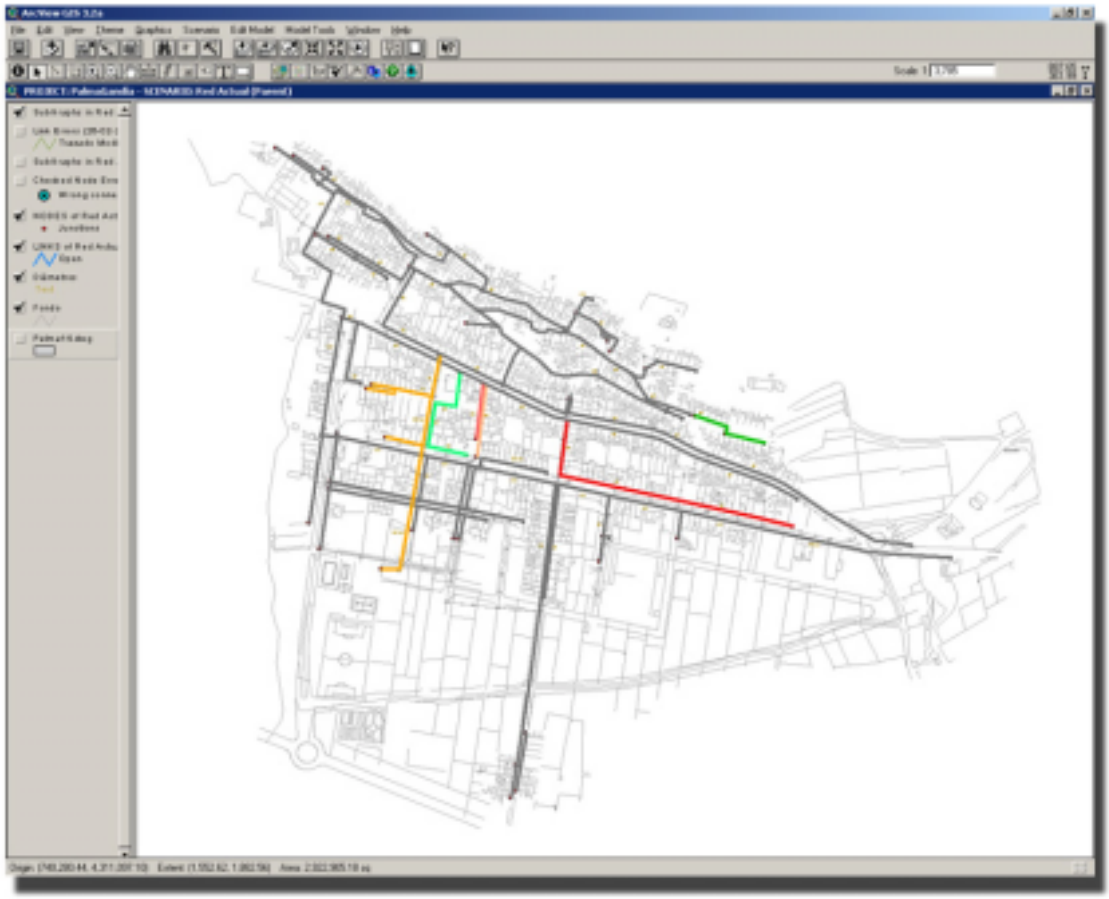


Fig. 66 Checking the network connectivity. Number of Sub-networks.

➢ To get one of the many **Trees** from the actual network:

1. Select **Model Tools | Connectivity**.
2. Select **Connectivity** from the emerging dialog.
3. Press **OK**.
4. A new theme will be created and added to the scenario classified by the number of subgraphs (sub-networks).
5. Change the classification of the theme:
    a. Edit the legend (for example double-clicking on the theme legend)
    b. In the legend editor, choose as **Legend Type 'Unique Value'**.
    c. Choose as **Values Field** the field **'T'**.
    d. The default classification means: **C = co-tree; T = tree.**
    e. Change the symbols and labels at will.
    f. Press the **Apply** button.

The next figure shows an example of a network classified by pipes that belong to the tree and pipes that belong to the co-tree, according to the *Graph Theory*.

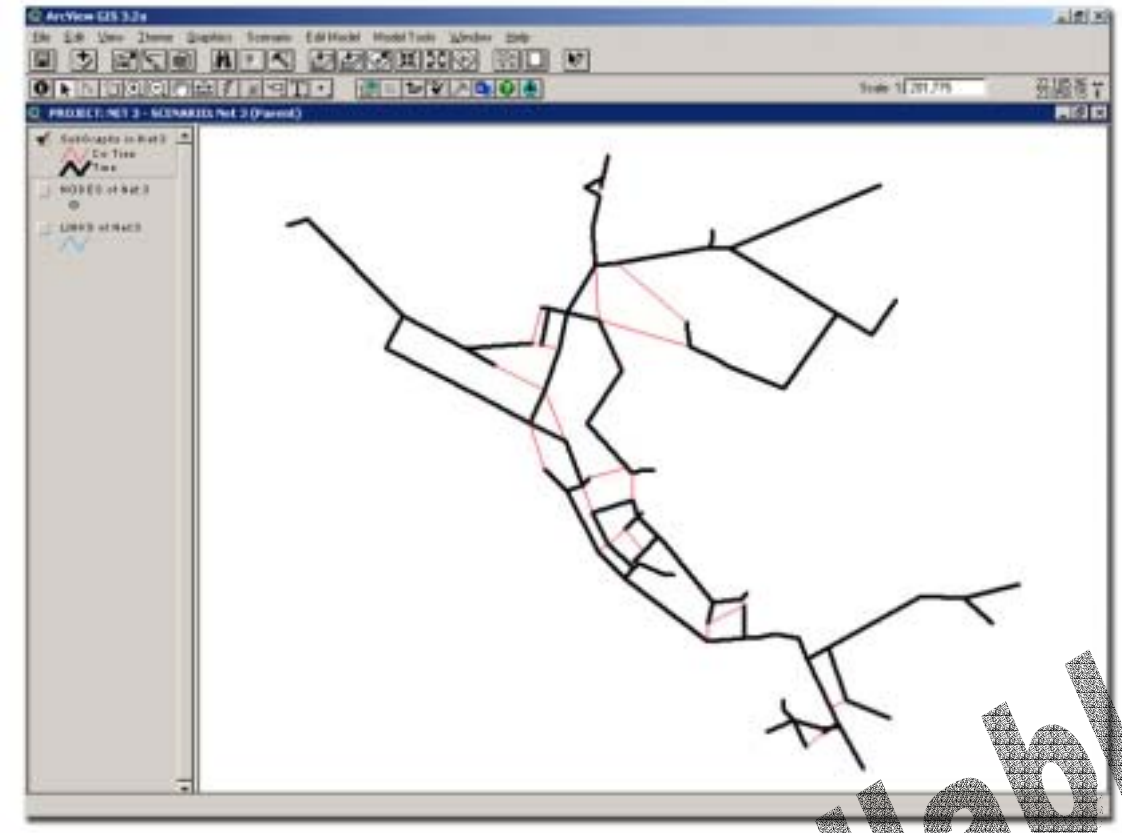


Fig. 67 Network tree and co-tree.

➢ To get the **shortest spanning tree**:

1. Select **Model Tools | Connectivity**.
2. Select **Shortest Spanning Tree** from the emerging dialog.
3. Select the **hydraulic property** the spanning tree is going to be based on (*Resistance*, *Diameter or Flow*).
4. Press **OK**.
5. A new theme will be created and added to the scenario specifying the shortest spanning tree.

Still Not Available

CHAPTER 13. ELEVATION INTERPOLATION

The elevation of a junction is always a required property when trying to run a model. Elevation is used only to compute pressure at the junction, thus, it is a basic parameter in order to obtain reliable results. Most GIS packages support different algorithms to make an interpolation surface from a set of points with an elevation reference in the corresponding attribute table. They can also support DTM (digital terrain models) from which the elevation at nodes can be directly computed. GISRed can interpolate elevations at nodes from a theme of scattered points making use of the IDW (*Inverse distance weighted*) method. It is also possible to make use of a GRID theme to interpolate elevations at each node, in this way, the Spatial Analyst extension is required.

➢ The process to compute the elevation at each node using a theme of **scattered points (IDW method)** is as follows:

1. Select **Model Tools** | **Elevation Interpolation**.
2. The following dialog appears:

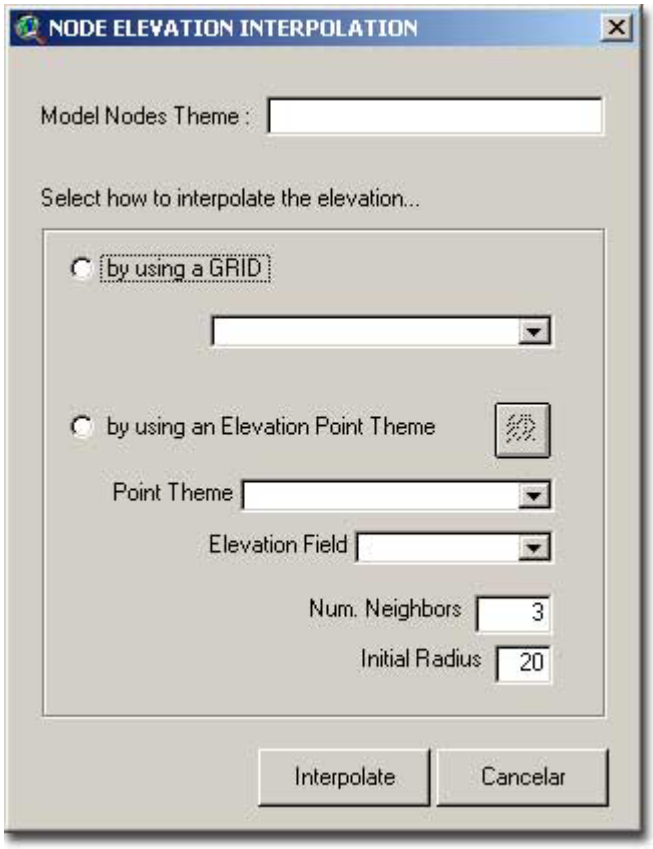


Fig. 68- Elevation Interpolation Dialog.

The first field within the dialog is non editable and corresponds to the node theme of the network.

3. Choose the option 'by using an Elevation Point Theme'.
4. Select the point theme that contains the elevations.
5. Specify the elevation field.

6. Specify the parameters for the interpolation method: the number of *Nearest Neighbors* and the *initial Radius* in the interpolation. The number of Nearest Neighbors is the total number of input points; the initial Radius is the starting radius to search for points.
7. Press **Interpolate**.

Automatically the elevations at **selected** network nodes are computed. If there is no selection, elevations are computed at all nodes.

➢ Making use of a **GRID theme**, the process should be the following:

To generate the GRID theme:

1. Load the Spatial Analyst extension. **File | Extensions** and toggle on the **Spatial Analyst** extension.
2. Open a view document (with the *ArcView Project Window* active, double-click the *View Icon* or alternatively, press ***New*** when the *View Icon* is active).
3. Add the theme that contains the set of points with elevation information. **View | Add Theme** or click on .

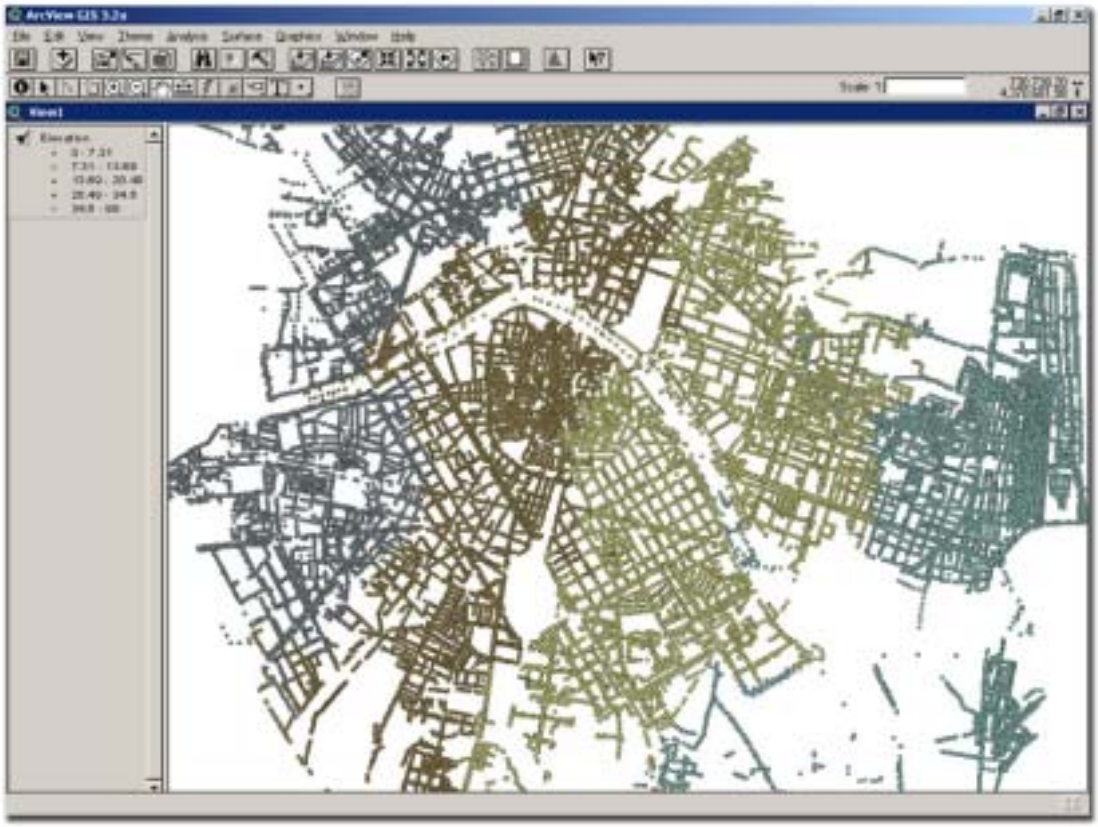


Fig. 69 Elevation Interpolation. Theme of Elevations.

4. Select **Surface | Interpolation Grid** from the menu bar. The user will have to go through a couple of interpolation surface dialogs.
5. Accept the defaults for the first one.

6. In the second one, choose the **IDW** (*Inverse Distance Weighted*) method and the **field that contains the elevation information (*Z Value Field*)**. Accept the rest of the fields.
7. Press **OK** to generate the surface.

The surface generated is showed in a temporary theme. It is possible to work with the temporary theme or generate a permanent Grid. In this case, a permanent theme is to be created. [*Skip the next three steps and go to 11 if you decide to work with the temporary theme*].

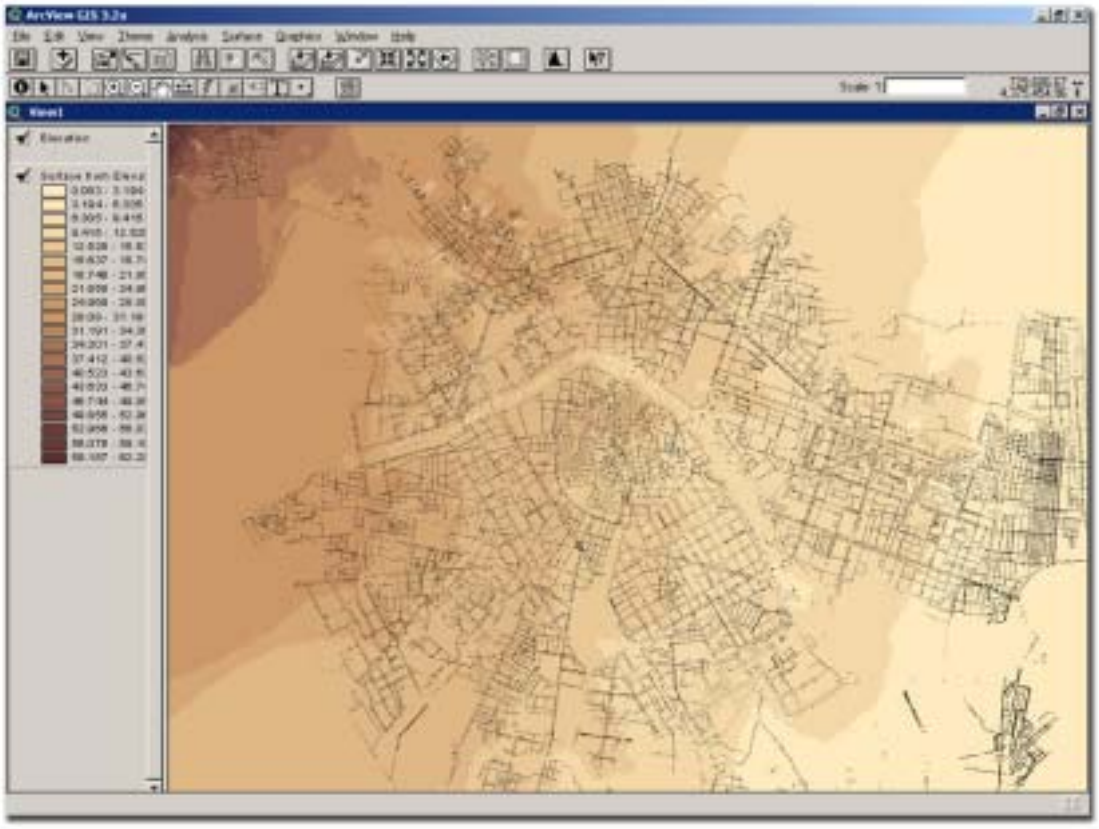


Fig. 70 Elevation Interpolation. Interpolation Surface.

8. To create a permanent grid, make active the temporary surface that was generated and select **Theme | Convert to Grid**.
9. Give a name ('*Grid*' for instance) and a path for the theme.
10. A new grid will be created. Press **Yes**, when ArcView asks you to add the theme to the view.

Now, the surface theme must be copied to the scenario before going forward to the interpolation process.

11. Make active the surface theme in the view.
12. Select **Edit | Copy Themes** from the menu bar.
13. Activate the scenario window.
14. Select **Edit | Paste**.

Once the GRID is generated, the GISRed Elevation Interpolation dialog can be used:

15. Select **Model Tools | Elevation Interpolation**.

16. The following dialog appears:

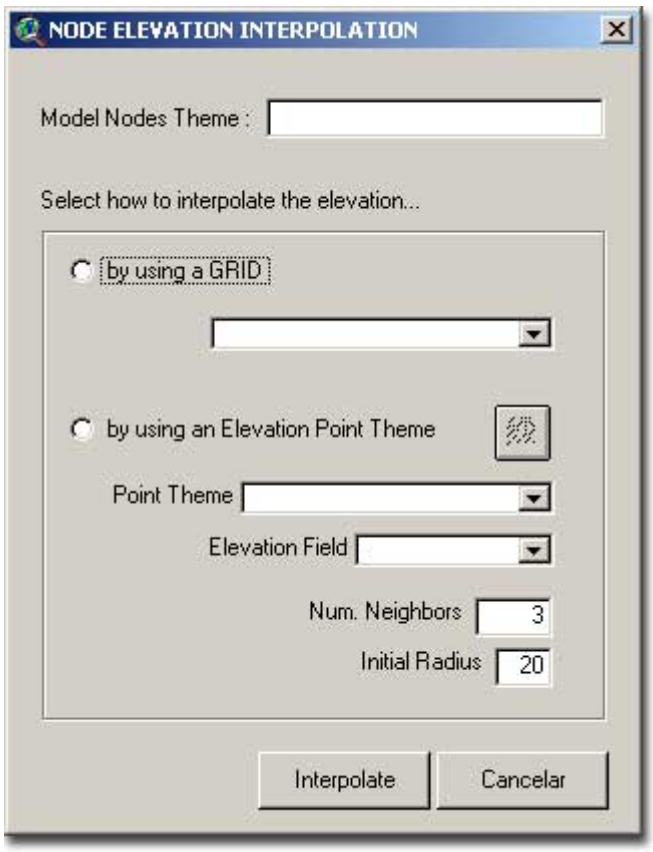


Fig. 71 Elevation Interpolation Dialog.

The first field within the dialog is non editable and corresponds to the node theme of the network.

17. Choose the grid theme to use in the second field.
18. Press **Interpolate**.

Automatically the elevations at each network node are calculated based upon the interpolation surface.

The user can check the elevation values once the process is finished, by editing the properties of the nodes.

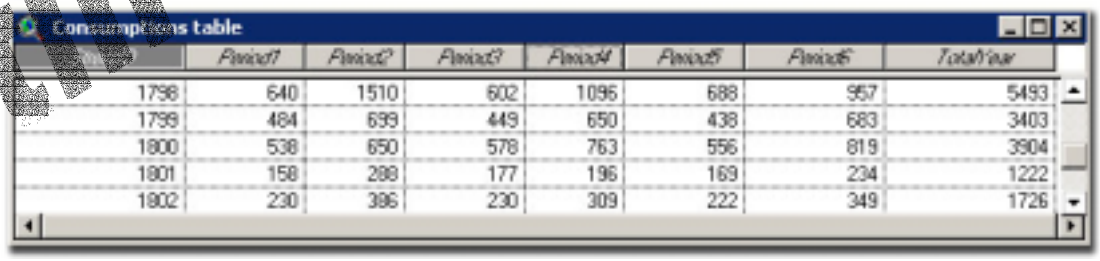
CHAPTER 14. DEMAND ALLOCATION

The demand allocation in a water distribution system is usually one of the most important and critical tasks when modeling a real network, to such an extent that it is customary to take advantage of the capabilities of GIS packages to cope with it. Since this process is mainly based on spatial operations, it is very common to use external tables containing consumption data in conjunction with the spatial information available in the GIS. In this regard, GISRed introduces a customized demand allocation tool to assist in the distribution of the total demand throughout the network.

The demand allocation tool automatically assigns metered consumptions grouped together by streets to the nearest demand node in the water distribution network. The demand nodes that are going to be taken during the process are nodes which belong to pipes with a diameter lower than a specified value.

Things that are needed before getting on with it:

- **Theme of streets** (same structure as the '***Road Segments***' custom-built theme of the *custom-built theme manager*).
- **Table of metered consumptions grouped by streets**. It is a dbf table specifying in one field (called **IdCalle** or **StreetID**) the *Street Identifier* (number), and in the rest of the fields either the *total consumption per year* or the *consumption every 2 months* (bimonthly consumption). **The consumption must be in cubic meters (m^3).**



Period = 2 months

Fig. 72 Demand Allocation. Consumption Table.

- **Register the table** of consumptions into the *Master Table*.

➢ To begin with the **demand allocation process**:

1. Register the table of consumptions:
   a. Open the master table and start editing the table.
   b. Add a new record specifying at least the following fields:
      i. *NAME*: a name for the demand table.
      ii. *CATEGORY*: DEMAND
      iii. *PATH*: the path of the table on disk.
   c. Stop editing and save edits.

2. Once the demand table has been registered, select **Model Tools | Demand Allocation**. The demand allocation dialog will appear.

Still Not Available

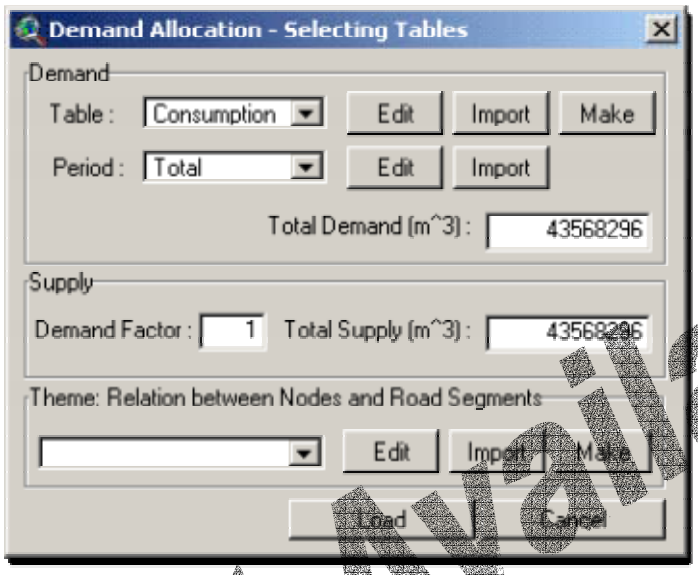


Fig. 73 Demand Allocation Dialog.

3. Adjust either the **demand factor** or the **total supply** (changing one of the fields will automatically adjust the other) in order to balance the volume injected to the system and the total consumption.

4. Choose the theme that relates Nodes and Road Segments. Alternatively, press **Make** to create a new one. Note that a Road Segment theme is required to be in the TOC before going on.
   a. If the button **Make** was pressed the following dialog will show up:

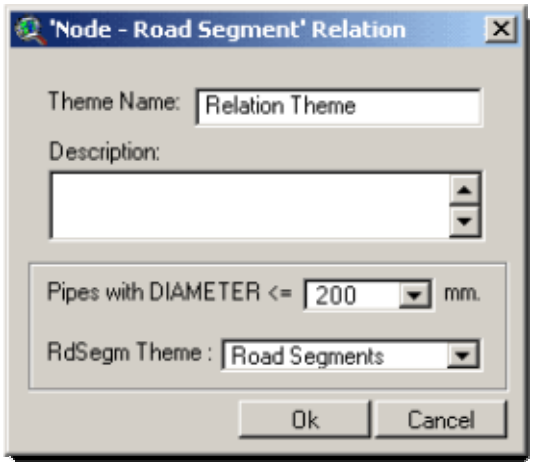


Fig. 74 Demand Allocation. Node-Road Segment Relation Theme.

   b. Enter a **name** for the relation theme.
   c. Specify the maximum **diameter** to consider when choosing consumption nodes.
   d. Specify the **Road Segment Theme**.
   e. Press **OK**.
   f. A New relation theme will be added to the TOC and the demand allocation dialog will be opened again.

5. Adjust the **demand factor** or the **total supply** if necessary.

Still Not Available

6. Press **Load** to allocate demands.

7. After the process, a summary window will come up.

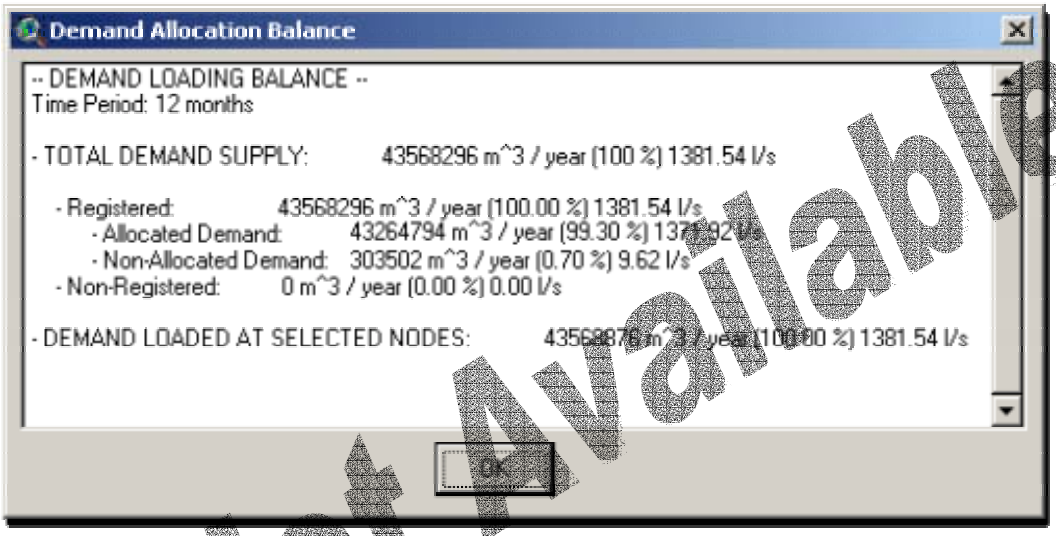

Demand Allocation Balance
-- DEMAND LOADING BALANCE --
Time Period: 12 months
- TOTAL DEMAND SUPPLY: 43568296 m^3 / year (100 %) 1381.54 l/s
- Registered: 43568296 m^3 / year (100.00 %) 1381.54 l/s
- Allocated Demand: 43264794 m^3 / year (99.30 %) 1371.92 l/s
- Non-Allocated Demand: 303502 m^3 / year (0.70 %) 9.62 l/s
- Non-Registered: 0 m^3 / year (0.00 %) 0.00 l/s
- DEMAND LOADED AT SELECTED NODES: 43568876 m^3 / year (100.00 %) 1381.54 l/s
OK

Fig. 75 Demand Allocation. Report Window.

Still Not Available

CHAPTER 15. ANALYSIS OPTIONS

After a network has been suitably described, its hydraulic and water quality behaviour can be analyzed. This chapter describes how to specify options to use in the analysis.

15.1 SETTING ANALYSIS OPTIONS

There are five categories of options that control how EPANET analyzes a network and that are reproduced in GISRed in three categories:

Hydraulics, Quality-Reactions and Times-Energy. To set any of these options:

1. Select the **Scenario | Analysis Options** from the menu bar.
2. Select **Hydraulics, Quality-Reactions or Times-Energy** from the category list.
3. The contents of the dialog change depending on the category.
4. Edit your option choices in the Analysis Options Dialog.

As you are editing a category of options in the Analysis Options Dialog, you can move to another category by simply choosing another one in the *category list*.

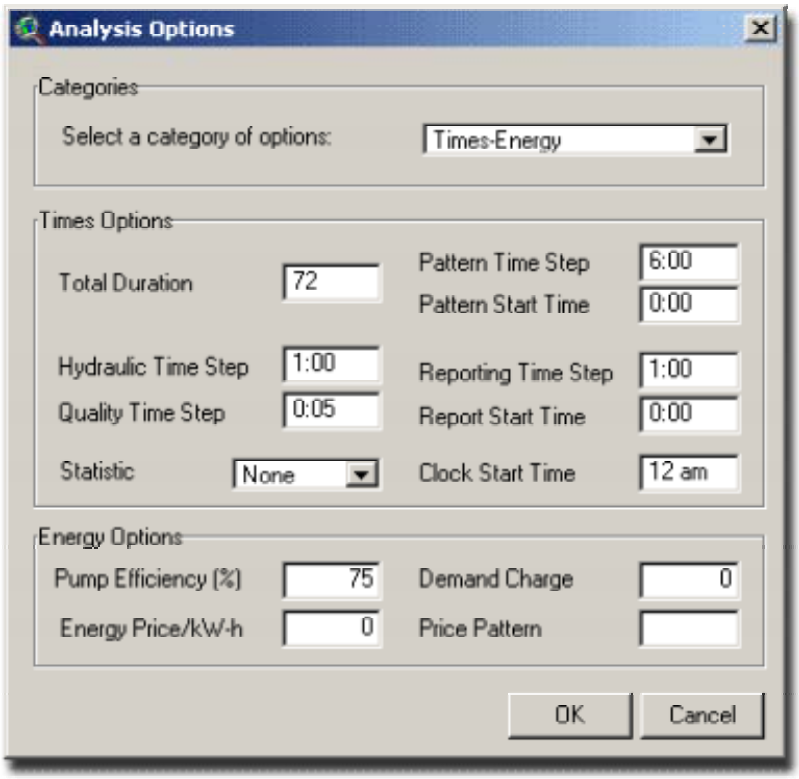


Fig. 76 Analysis Options Dialog.

15.1.1 Hydraulic Options

Hydraulic options control how the hydraulic computations are carried out. They consist of the following items:

OPTION	DESCRIPTION
Flow Units	Units in which nodal demands and link flow rates are expressed. Choosing units in gallons, cubic feet, or acre-feet implies that the units for all other network quantities are Customary US. Selecting liters or cubic meters causes all other units to be SI metric. Use caution when changing flow units as it might affect all other data supplied to the project. (See Appendix B, Units of Measurement.)
Headloss Formula	Formula used to compute headloss as a function of flow rate in a pipe. Choices are: •Hazen-Williams •Darcy-Weisbach •Chezy-Manning Because each formula measures pipe roughness differently, switching formulas might require that all pipe roughness coefficients be updated.
Specific Gravity	Ratio of the density of the fluid being modeled to that of water at 4 deg. C (unitless).
Relative Viscosity	Ratio of the kinematic viscosity of the fluid to that of water at 20 deg. C (1.0 centistokes or 0.94 sq ft/day) (unitless).
Maximum Trials	Maximum number of trials used to solve the nonlinear equations that govern network hydraulics at a given point in time. Suggested value is 40.
Accuracy	Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the sum of all flow changes divided by the sum of all link flows is less than this number. Suggested value is 0.001.
If Unbalanced	Action to take if a hydraulic solution is not found within the maximum number of trials. Choices are STOP to stop the simulation at this point or CONTINUE to use another 10 trials, with no link status changes allowed, in an attempt to achieve convergence.
Default Pattern	ID label of a time pattern to be applied to demands at those junctions where no time pattern is specified. If no such pattern exists then demands will not vary at these locations.
Demand Multiplier	Global multiplier applied to all demands to make total system consumption vary up or down by a fixed amount. E.g., 2.0 doubles all demands, 0.5 halves them, and 1.0 leaves them as is.
Emitter Exponent	Power to which pressure is raised when computing the flow through an emitter device. The textbook value for nozzles and sprinklers is ½. This may not apply to pipe leakage. Consult the discussion of

	Emitters in Section 3.1 for more details.
Status Report	Amount of status information to report after an analysis is made. Choices are: •NONE (no status reporting) •YES (normal status reporting – lists all changes in link status throughout the simulation) •FULL (full reporting – normal reporting plus the convergence error from each trial of the hydraulic analysis made in each time period) Full status reporting is only useful for debugging purposes.

15.1.2 Water Quality Options

Water Quality Options control how the water quality analysis is carried out. They consist of the following:

OPTION	DESCRIPTION
Parameter	Type of water quality parameter being modeled. Choices include: • NONE (no quality analysis), • CHEMICAL (compute chemical concentration), • AGE (compute water age), • TRACE (trace the percent of flow originating from a specific node). In lieu of CHEMICAL, you can enter the actual name of the chemical being modeled (e.g., Chlorine).
Mass Units	Mass units used to express concentration. Choices are mg/L or μg/L. Units for Age and Trace analyses are fixed at hours and percent, respectively.
Trace Node	ID label of the node whose flow is being traced. Applies only to flow tracing analyses.
Relative Diffusivity	Ratio of the molecular diffusivity of the chemical being modeled to that of chlorine at 20 deg. C (0.00112 ft^2/day). Use 2 if the chemical diffuses twice as fast as chlorine, 0.5 if half as fast, etc. Applies only when modeling mass transfer for pipe wall reactions. Set to zero to ignore mass transfer effects.
Quality Tolerance	Smallest change in quality that will cause a new parcel of water to be created in a pipe. A typical setting might be 0.01 for chemicals measured in mg/L as well as water age and source tracing.

Note: *The Quality Tolerance determines when the quality of one parcel of water is essentially the same as another parcel. For chemical analysis this might be the detection limit of the procedure used to measure the chemical, adjusted by a suitable factor of safety. Using too large a value for this tolerance might affect simulation accuracy. Using too small a value will affect computational efficiency. Some experimentation with this setting might be called for.*

15.1.3 Reaction Options

Reaction Options set the types of reactions that apply to a water quality analysis. They include the following:

OPTION	DESCRIPTION
Bulk Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored.
Wall Reaction Order	Power to which concentration is raised when computing a bulk flow reaction rate. Choices are (1) for first-order reactions or (0) for constant rate reactions. If no global or pipe-specific wall reaction coefficients are assigned then this option is ignored.
Global Bulk Coefficient	Default bulk reaction rate coefficient (K_b) assigned to all pipes. This global coefficient can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no bulk reaction occurs. Units are concentration raised to the (1-n) power divided by days, where n is the bulk reaction order.
Global Wall Coefficient	Wall reaction rate coefficient (K_w) assigned to all pipes. Can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no wall reaction occurs. Units are ft/day (US) or m/day (SI) for first-order reactions and mass/sq ft/day (US) or mass/sq m/day (SI) for zero-order reactions.
Tank Reaction Order	It is used to set the order of reactions occurring in the tanks. Use 1 for first-order reactions, 2 for second-order reactions, etc. If not supplied the default reaction order is 1.0.
Wall Coefficient Correlation	Factor correlating wall reaction coefficient to pipe roughness. Set to zero if not applicable.
Limiting Concentration	Maximum concentration that a substance can grow to or minimum value it can decay to. Bulk reaction rates will be proportional to the difference between the current concentration and this value. Set to zero if not applicable.

15.1.4 Times Options

Times options set values for the various time steps used in an extended period simulation. These are listed below (times can be entered as decimal hours or in *hours:minutes* notation):

OPTION	DESCRIPTION
Total Duration	Total length of a simulation in hours. Use 0 to run a single period (snapshot) hydraulic analysis.
Hydraulic Time Step	Time interval between re-computation of system hydraulics. Normal default is 1 hour.
Quality Time Step	Time interval between routing of water quality constituent. Normal default is 5 minutes (0:05 hours).
Pattern Time Step	Time interval used with all time patterns. Normal default is 1 hour.
Pattern Start Time	Hours into all time patterns at which the simulation begins (e.g., a value of 2 means that the simulation begins with all time patterns starting at their second hour). Normal default is 0.
Reporting Time Step	Time interval between times at which computed results are reported. Normal default is 1 hour.
Report Start Time	Hours into simulation at which computed results begin to be reported. Normal default is 0.
Clock Start Time	Clock time (e.g., 7:30 am, 10:00 pm) at which simulation begins. Default is 12:00 am (midnight).
Statistic	Type of statistical processing used to summarize the results of an extended period simulation. Choices are: • NONE (results reported at each reporting time step) • AVERAGE (time-averaged results reported) • MINIMUM (minimum value results reported) • MAXIMUM (maximum value results reported) • RANGE (difference between maximum and minimum results reported) Statistical processing is applied to all node and link results obtained between the Report Start Time and the Total Duration.

Note: *To run a single-period hydraulic analyses (also called a snapshot analysis) enter 0 for Total Duration. In this case entries for all of the other time options, with the exception of Starting Time of Day, are not used. Water quality analyses always require that a non-zero Total Duration be specified.*

15.1.5 Energy Options

Energy Analysis Options provide default values used to compute pumping energy and cost when no specific energy parameters are assigned to a given pump. They consist of the following:

OPTION	DESCRIPTION
Pump Efficiency (%)	Default pump efficiency.
Energy Price per Kw-h	Price of energy per kilowatt-hour. Monetary units are not explicitly represented.
Price Pattern	ID label of a time pattern used to represent variations in energy price with time. Leave blank if not applicable.
Demand Charge	Additional energy charge per maximum kilowatt usage.

CHAPTER 16. NETWORK SIMULATION

16.1 EXPORTING EPANET INPUT FILES

To export an EPANET input file:

1. Select **Scenario | Export Epanet Input File** from the menu bar or click on the GISRED toolbar.
2. Choose a directory to save the file and enter a name for the Epanet Input File.
3. Click **OK**.

The Epanet input file is generated. Now, you can open this file in EPANET and run a simulation.

16.2 RUNNING AN ANALYSIS

To run a hydraulic/water quality analysis:

1. Select **Scenario | Run Analysis** from the menu bar.
2. The user will be asked to choose an EPANET input file to run.
3. Once chosen, the progress of the analysis will be displayed in the status bar.
4. At the end of the analysis a report window will appear.

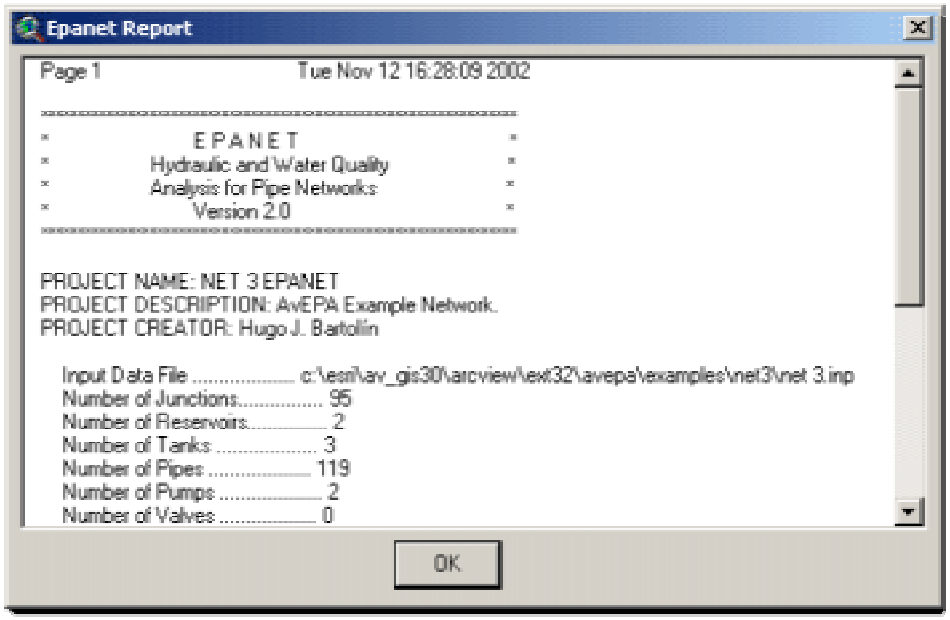

Epanet Report
Page 1 Tue Nov 12 16:28:09 2002
EPANET
Hydraulic and Water Quality
Analysis for Pipe Networks
Version 2.0
PROJECT NAME: NET 3 EPANET
PROJECT DESCRIPTION: AvEPA Example Network.
PROJECT CREATOR: Hugo J. Bartolin
Input Data File ..... c:\esri\av_gis30\arcview\ext32\avepa\examples\net3\net 3.inp
Number of Junctions..... 95
Number of Reservoirs..... 2
Number of Tanks 3
Number of Pipes 119
Number of Pumps 2
Number of Valves 0
OK

Fig. 77 Epanet Report Window.

5. Click **OK** to display the browser.
6. The Browser window will appear.

16.3 VIEWING RESULTS ON THE SCENARIO

It is possible to view database values and results of a simulation directly on the Network Scenario.

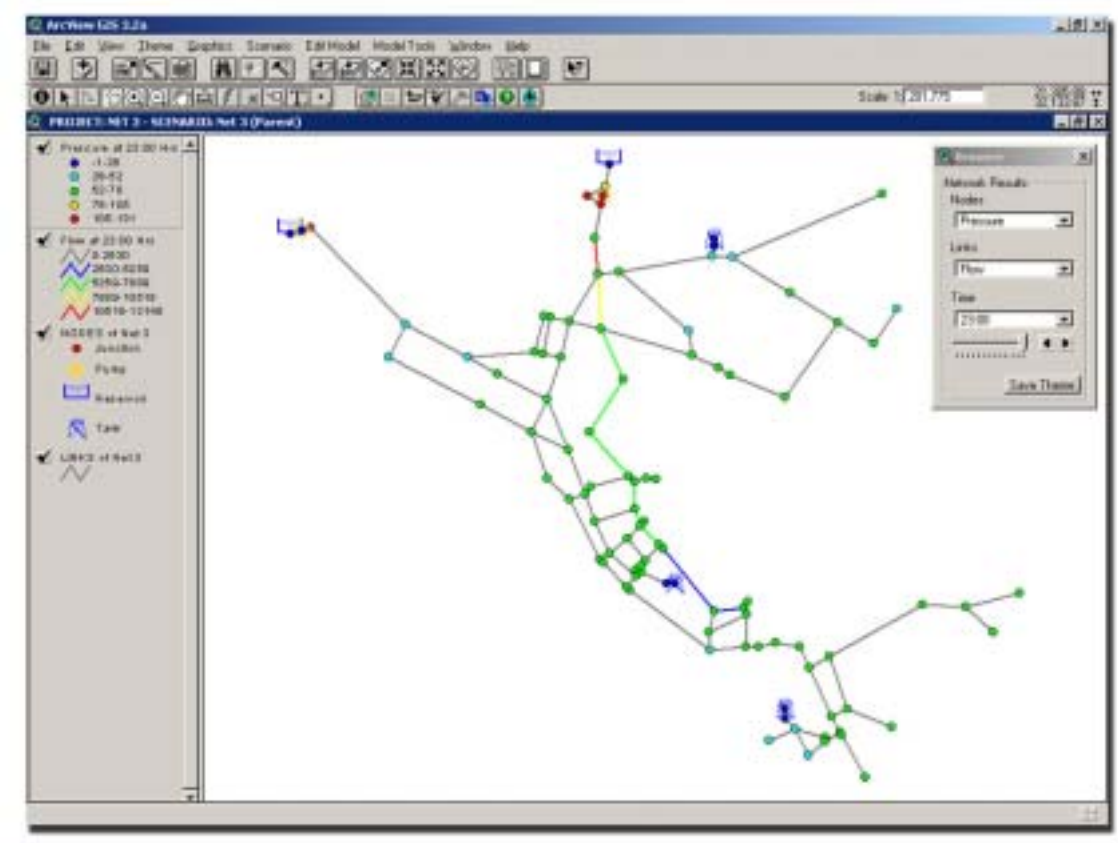
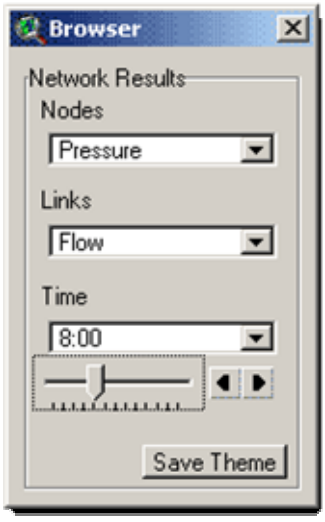


Fig. 78 Viewing Results on the Scenario.



Using the Browser of Results, the user can choose a parameter for nodes (*Demand, Head, Pressure and Quality*) and a parameter for links (*Flow, Velocity and Unit Headloss*) to be graphically represented at a specific period of time on the scenario. A new theme of nodes is added to the TOC when the user chooses any node parameter. If the user changes the parameter for nodes, the theme will be automatically updated with the new parameter and values. Additionally, the theme is updated with the results of the period of time chosen either with the time list or with the slider. The same applies for link parameters. Note that these themes are temporary themes (Clones of the original node and link themes but Classified by a chosen parameter at a period of time).

To hide the result theme, choose the ***No View*** option in the list of parameters, automatically the application will remove the corresponding theme from the TOC.

To make a theme of results a permanent theme, press the **Save Theme button** and give a name for the theme in the window that appears. The scenario's colouring will be updated as a new time period is selected in the Browser slider control.

To close the browser click the ☒ button of the browser window.

If a simulation has been run and the browser has been closed, select **Scenario | Browser of Results** to open the browser again.

CHAPTER 17. THE NETWORK MODEL CALIBRATION

Calibration of pipe network systems consists of determining the physical and operational characteristics of an existing system. This is achieved by determining various parameters that, when input into a hydraulic simulation model, will yield a reasonable match between measured and predicted pressures and flows in the network (Shamir et al., 1968).

17.1 INTRODUCTION

In order to illustrate the capability of the calibration module of the **GISRed Extension,** the following example is presented. The network to calibrate is a small water distribution system of a hypothetical community, Anytown (USA), originally introduced by Walski et al. in 1987. The network layout is shown in the following picture:

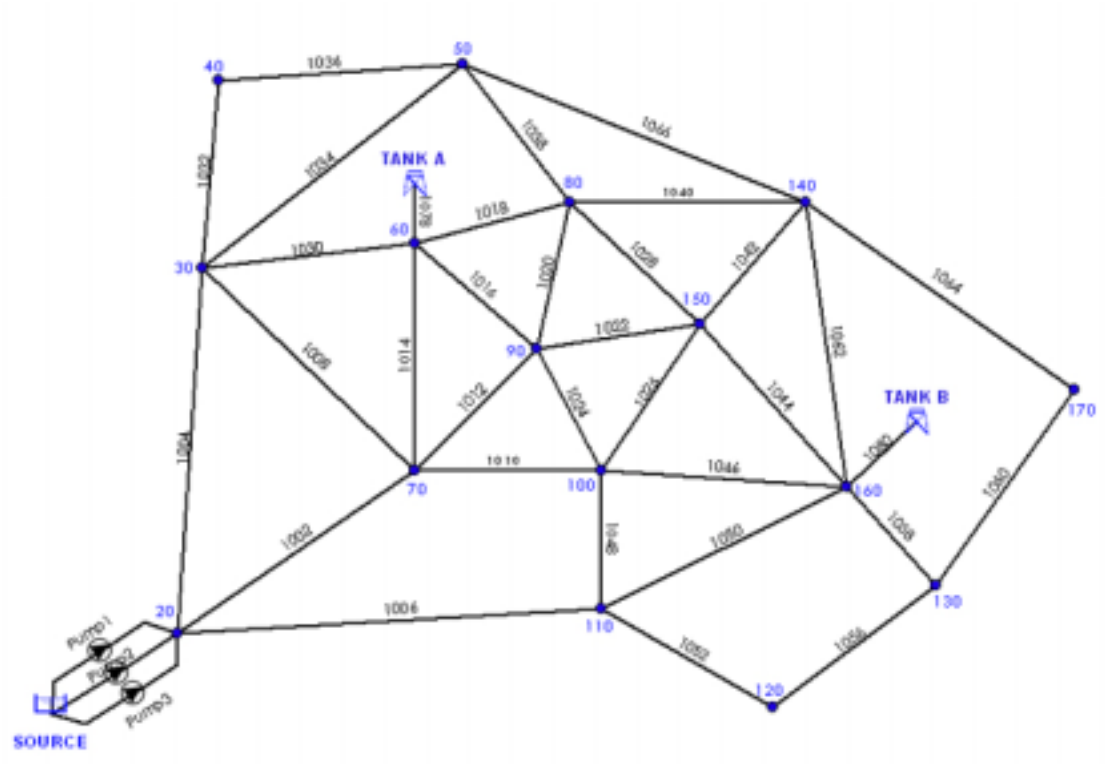


Fig. 79 Anytown Network Layout.

17.2 NETWORK PROPERTIES

The water distribution system consists of 40 pipes, 22 junction nodes (16 demand nodes, 6 non-demand nodes), two elevated tanks, one fixed head source (reservoir) and three pumps. The complete network characteristics are given below. For this example, pipe roughness was expressed in terms of the Hazen-Williams C factor. Both tanks have bottom elevations of 65.5 m and overflow elevations of 79.5 m. the water level in the clear-well is maintained at an elevation of 3.04 m. All three pumps have identical pump characteristic curves. A **unique demand pattern** is considered for all demand nodes. **Four monitoring points** located on nodes **40**, **90**, **120** and **140** recorded head measurements during a hypothetical field test. (Ormsbee, 1989)

NODES

NODE	ELEVATION (m)	DEMAND (l/s)
20	6,23	31,51
30	15,24	12,52
40	15,24	12,52
50	15,24	31,51
60	15,24	50,90
70	15,24	31,51
80	15,24	31,51
90	15,24	63,88
100	15,24	12,52
110	15,24	12,52
120	36,60	31,51
130	36,60	12,52
140	24,40	12,52
150	36,60	12,52
160	36,60	31,51
170	36,60	12,52
TOTAL		**404,00**

**Red numbers indicate monitoring points | LC = Loading Condition*

PIPES

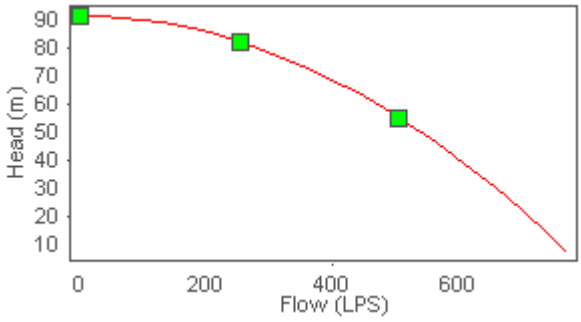
PIPE	NODE 1	NODE 2	LENGTH (m)	DIAMETER (mm)	ACTUAL C-FACTOR (H-W)	ASSUMED C-FACTOR (H-W)	GROUP
1002	20	70	3657	406	120	110	1
1004	20	30	3657	406	120	110	1
1006	20	110	3657	406	120	110	1
1008	30	70	2743	305	70	110	2
1010	70	100	1830	305	120	110	1
1012	70	90	1830	254	70	110	2
1014	70	60	1830	305	70	110	2
1016	60	90	1830	254	70	110	2
1018	60	80	1830	305	70	110	2
1020	80	90	1830	254	70	110	2
1022	90	150	1830	254	70	110	2
1024	90	100	1830	254	70	110	2
1026	100	150	1830	305	70	110	2
1028	80	150	1830	254	90	110	3
1030	30	60	1830	254	120	110	1
1032	30	40	1830	254	120	110	1
1034	30	50	2743	254	120	110	1
1036	40	50	1830	254	120	110	1
1038	50	80	1830	254	120	110	1
1040	80	140	1830	254	130	110	4
1042	140	150	1830	203	130	110	4
1044	150	160	1830	203	90	110	3
1046	100	160	1830	305	90	110	3
1048	100	110	1830	203	90	110	3
1050	110	160	1830	254	90	110	3
1052	110	120	1830	203	90	110	3
1056	120	130	1830	203	130	110	4
1058	130	160	1830	254	130	110	4
1060	130	170	1830	203	130	110	4
1062	140	160	1830	203	130	110	4
1064	140	170	3656	203	130	110	4
1066	50	140	3656	203	130	110	4
1078	60	TANK A	31	305	110	110	5
1080	160	TANK B	31	305	110	110	5

PUMPS
For all loading conditions all three pumps are assumed to be in operation.

PUMP	PIPE	LENGTH (m)	DIAMETER (mm)	C-FACTOR (H-W)
PUMP 1	Suction Pipe	1	1600	150
	Discharge Pipe	1	1600	150
PUMP 2	Suction Pipe	1	1600	150
	Discharge Pipe	1	1600	150
PUMP 3	Suction Pipe	1	1600	150
	Discharge Pipe	1	1600	150

Q-H CURVE (Identical curve for the 3 pumps)

Q-H CURVE	
Q (l/s)	H (m)
0,0	91,40
252,50	82,30
504,70	55,20



TANKS
Both tanks have identical characteristics and are filling and draining during a 24 hours simulation.

NODE	ELEVATION (m)	DIAMETER (m)	INITIAL LEVEL (m)	MAX. LEVEL (m)	MIN. LEVEL (m)
TANK A	65,5	21,55	6	14	0
TANK B	65,5	21,55	6	14	0

SOURCE (RESERVOIR)

NODE	ELEVATION (m)
SOURCE	3.04

DEMAND PATTERN

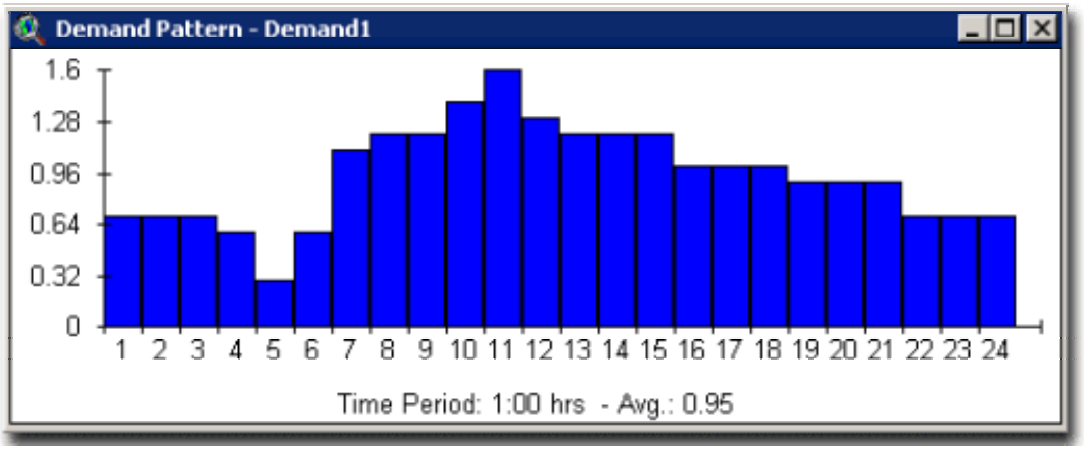
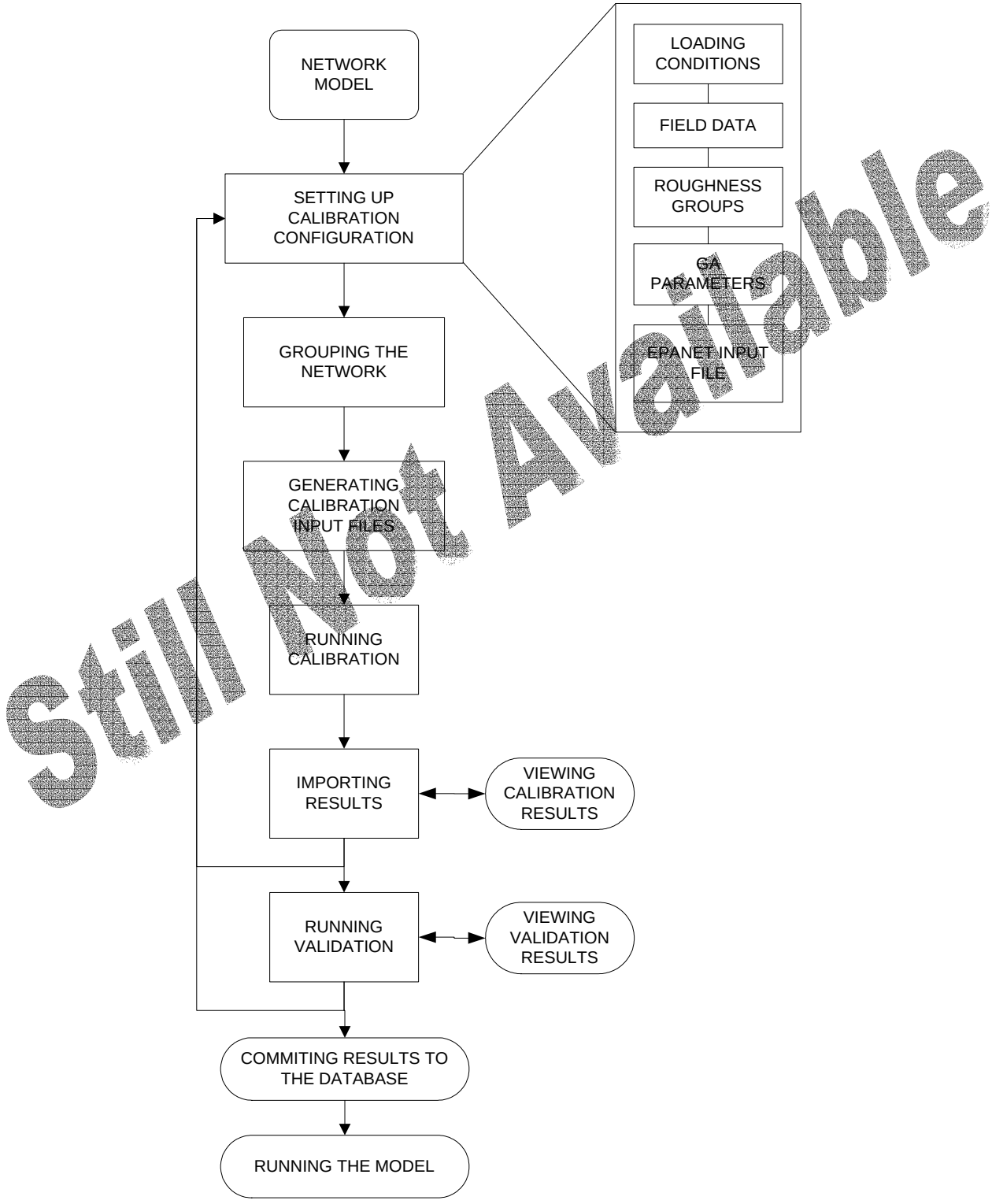


Fig. 80 Calibration Example. Demand Pattern.

17.3 CALIBRATION PROCESS

The following diagram depicts the process of calibration by means of the GISRed Extension.



Still Not Available

Fig. 81 GISRed Calibration Process. Flowchart.

Our first task is to open the **Anytown network example** if not opened yet.

1. Launch ArcView if it is not already running. Once opened, go to **Extensions** in the **File** menu when the ArcView Project Window is active and select **GISRED 1.0**.

2. Select **Network** | **Projects** to open the **Anytown** example. If the example is in the list of projects, select it and press open. If not, select **Add**, and enter the details required. *Remember that the example is provided along with the extension. So you should have a directory called Anytown in $AVEXT/GISRED/examples.*

Let us start up by defining a new calibration configuration:

1. Select **Model Tools** | **Calibration Administrator…** to see the main calibration dialog.

2. Press **New** and fill in all the required information. Enter a name for the Configuration and a path of a valid Epanet input file of the network you want to calibrate (you can always make one from the GISRed interface).

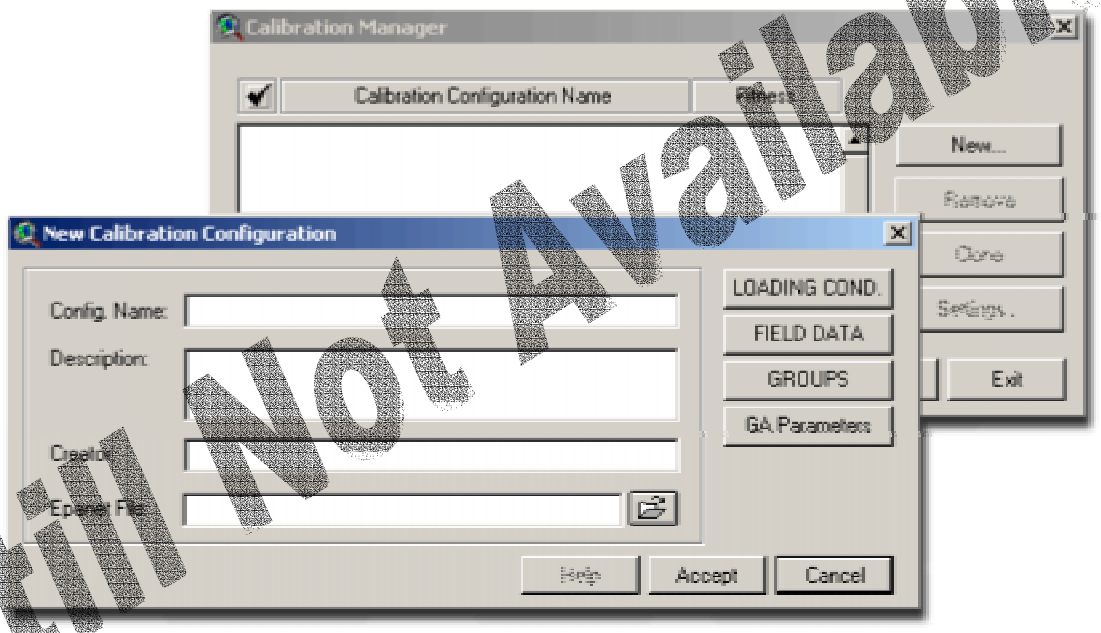


Fig. 82 New Calibration Configuration.

Still Not Available

Before accepting, we are going to enter the rest of the parameters required to make up the calibration configuration.

3. Now, select **Loading Cond.** to open the dialog form. In our example the duration of the simulation is 24 hours. Select all of them (by default if none of them is selected GISRed assumes all of them are to be included in the analysis). Note: *to select more than one hour, use the shift key.*

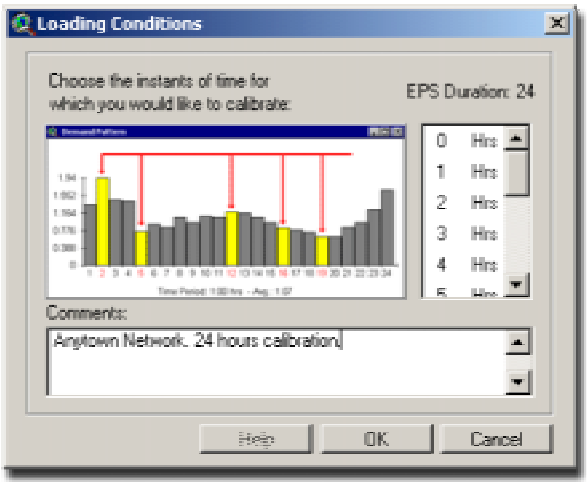


Fig. 83 Loading Conditions Dialog.

The next step is to specify the monitoring points that recorded the **Head Measurements**.

4. Press **Field Data**. Since there are just head measurements available, press Import (from the Head section). Fill in the gaps as shown in the next figure and press Import.

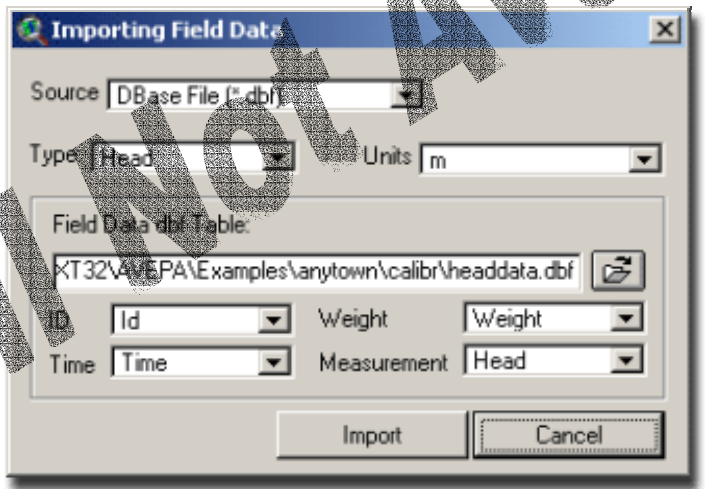


Fig. 84 Importing Field Data.

Still Not Available

After the importation, the main field data dialog should look like this:

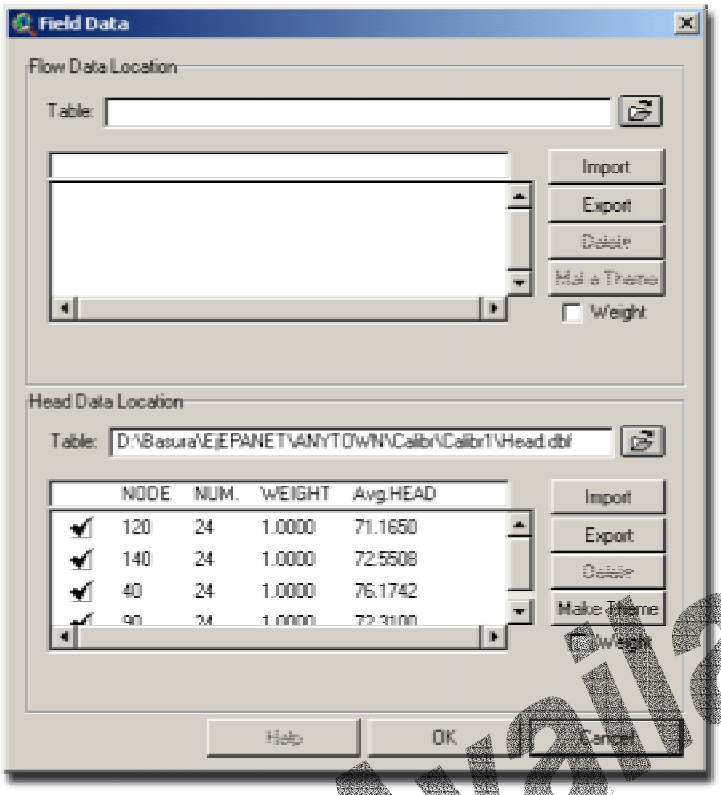


Fig. 85 Field Data after the importation.

Notice the check marks that appear beside each one of the measurement points. They allow the user to activate/deactivate the measurement. If deactivated, it means the measurement will be used for validation purposes. Press OK to accept all the changes.

Next step is to setup the Roughness Groups.

5. Select **GROUPS** in the New Calibration Configuration Dialog form. Define the following 5 decision variables:

GROUP ID	PRIOR ESTIMATE	Weight
1	140	0.00
2	60	0.00
3	89	0.00
4	135	0.00
5	100	0.00

***NOTE1**: We will group the network pipes later on, using the classification defined above.*
***NOTE2**: The weight of each group has been set up to 0, thus any prior estimate is neglected in the objective function.*

6. Next step is to define the **GA Parameters.**

Still Not Available

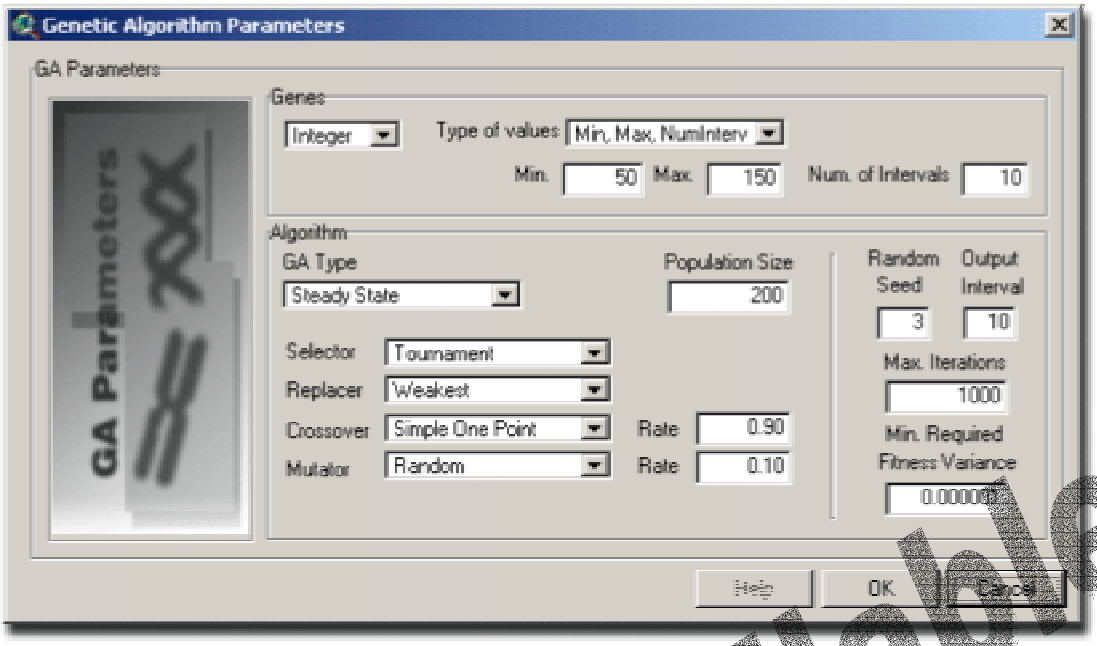


Fig. 86 GA Parameters dialog.

In this example we are going to use **Integer genes** that can take the values from within a finite interval (see Fig. 86). For the genetic algorithm the following parameters are selected:

GA Type	STEADY STATE
Selector	TOURNAMENT
Replacer	WEAKEST
Crossover	SIMPLE ONE POINT *Rate: 0.90*
Mutator	RANDOM *Rate: 0.10*
Population Size	200
Random Seed	3
Output Interval	10
Max. Iterations	1000
Min. Required Fitness Variance	1e-6

***NOTE**: To see an explanation of each one of the possible parameters, see the section titled 'The Standard Library' at the end of the example.*

7. Finally, the last step is to enter a valid Epanet Input File. If this file does not exist or changes were introduced in the actual model, a new file can be generated using the button of the GUI.

After setting all the parameters of the calibration configuration and accepting them, the Calibration Manager should look like this:

Still Not Available

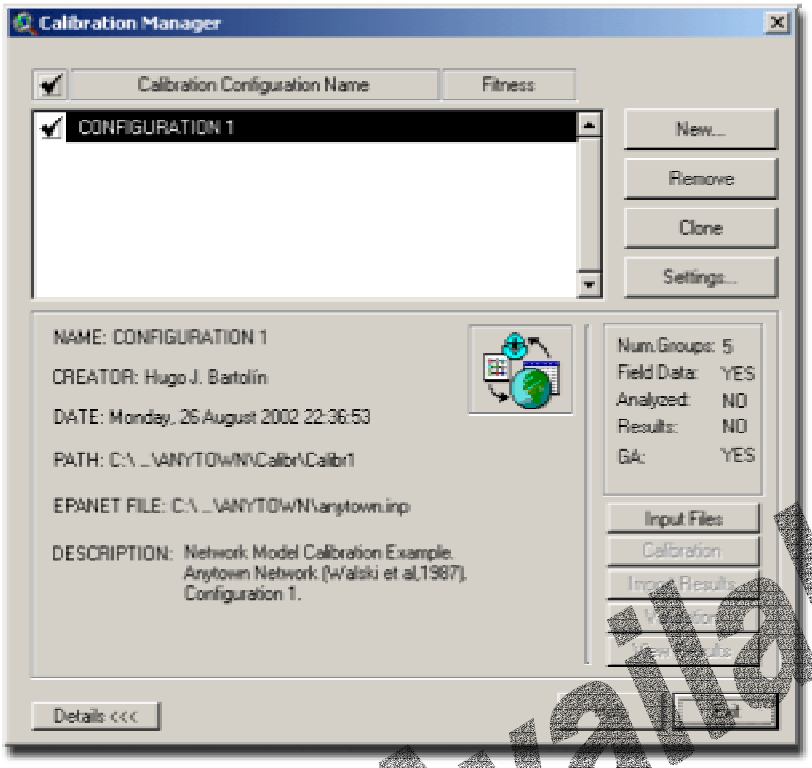


Fig. 87 Calibration Manager.

We have a **new calibration configuration**. The check mark beside the name of the configuration points out the current configuration. Since there might be several different configurations for the same network, the mark indicates the current one (it is obvious that only one configuration may be the current one at a time). By using the menu bar (see Fig. 88) instead of the calibration manager, the user will be prompted with the dialogs of the selected option for the current configuration.

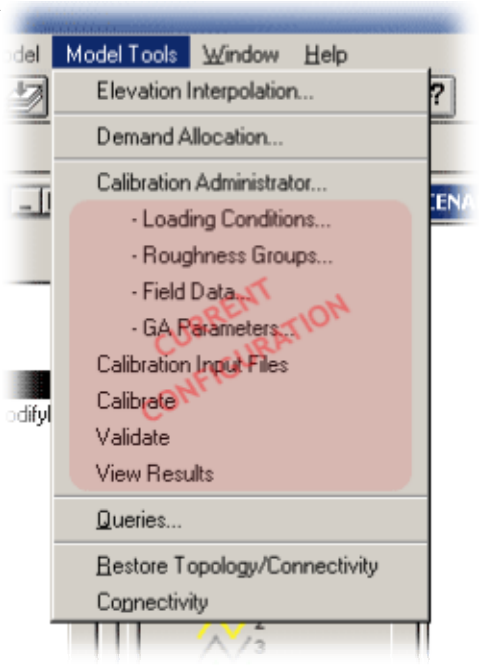


Fig. 88 Calibration menu bar options.

Still Not Available

Notice that only the button Input Files is active. But before going on with the creation of the calibration input files, the network must be grouped. Thus, exit the calibration manager and continue with the next section.

17.4 GROUPING THE NETWORK

At this point we are ready to group the network pipes. The grouping may be carried out in two possible ways:

1. By using the ArcView tool to select the pipes manually.

2. By means of particular queries. For this, you have to use the Query Builder tool that ArcView provides, . For instance, you might want to make a query to select all those pipes of the network whose diameter is lower than 300 and higher than 100. The syntaxis of the query would be: *(([Diameter] > 100) and ([Diameter] < 300))*

3. As the pipes are selected, assign them a group by using the **Group Edit** dialog.

    a. Bring up Edit Model | Group Edit…

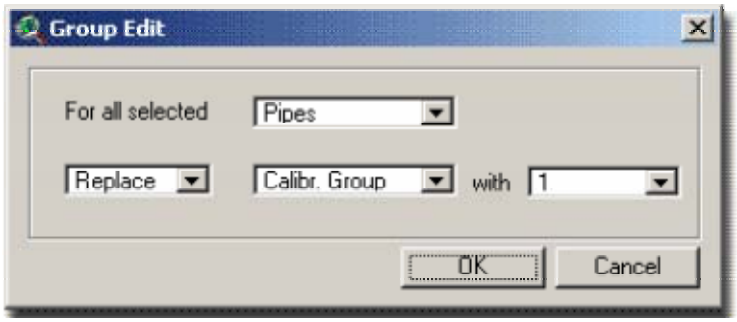


Fig. 89 Group Editing Dialog.

    b. Select the property Calibr.Group and the required Group ID from the dropdown list.

    c. Accept.

    GISRed shows a report window specifying the number of junctions affected by the change.

Alternatively and if further individual changes are required, the pipe property dialog can be used to assign or change the group which a pipe belongs to.

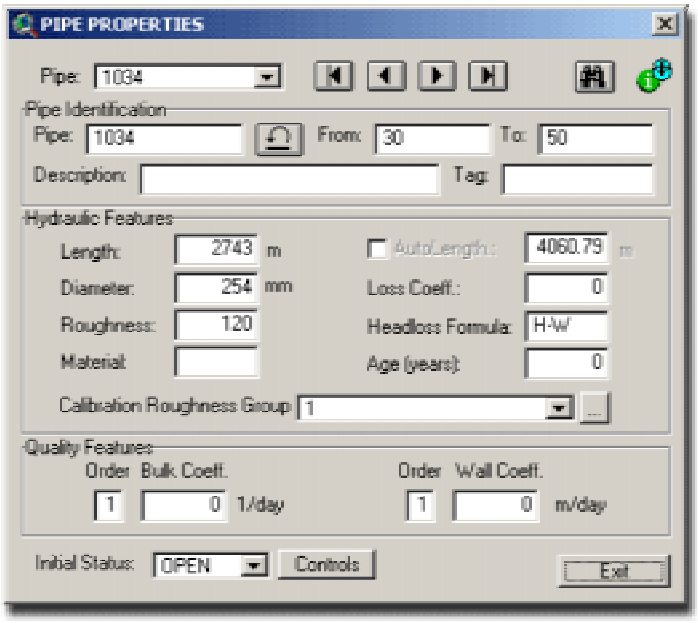


Fig. 90 Pipe Property Dialog. Changing Calibration Roughness Groups.

17.5 CALIBRATION INPUT FILES

Following the mainstream of the flowchart of the Fig. 81, the next step is the generation of the calibration input files. These files will allow the GA calibrator to do its job. To generate all the files needed, press the button '**Input Files**' of the calibration manager (see Fig. 87). Three text files are created in the current calibration directory, namely *main.in*, *lcdata.in* and *ga.in*.

17.6 RUNNING A CALIBRATION AND IMPORTING RESULTS

Once the three files are generated by the application, the button Calibration of the Calibration Manager will be activated. Press this button to launch the calibrator. A command window will appear and the calibrator will start running. When it gets to the end of the iterative process, three new files are generated, this time with *.out extension, namely main.out, HQ.out and ga.out. All these text files are imported into GISRed dbf tables.

Still Not Available

17.7 VIEWING CALIBRATION RESULTS

After the importation of all the calibration results, the fitness of the calibration process turns up. In this particular case, and for the GA parameters selected, the fitness appears to be 0.53020.

NOTE: *The objective function used by the calibrator is as follows (weighted sum of squared errors)*,

$$E = \sum_{i=1}^{N_{Heads}} \sum_{j=1}^{N_{LC}} w_{Hij} (H_{ij}^{*} - H_{ij})^2 + \sum_{i=1}^{N_{Flows}} \sum_{j=1}^{N_{LC}} w_{Qij} (Q_{ij}^{*} - Q_{ij})^2 + \sum_{k=1}^{N_{PE}} w_{\delta k} (\delta_k^{*} - \delta_k)^2$$

where:

E = Fitness (dimensionless)

H* = Measured Head

H = Predicted Head

Q* = Measured Flow

Q = Predicted Flow

δ^* = Prior estimate of the decision variable (*Pseudo Measurement*)

δ = Prior estimate of the decision variable

w = weights ($1/\sigma^2$), σ = generalized error (residual, in measurement units)

and the problem is formulated as a constrained optimization problem of weighted least square type, (that is, **Minimise E)**, subject to a particular set of nodal demands (loading conditions) and known boundary conditions (reservoir/tank heads, pump/valve status).

In this particular example the terms of flows and decision variable prior estimates are neglected, since we are not considering flow measurements and the weights for all decision variables are set to 0.

To see the entire set of results, press the button **View Results** in the **calibration manager**. The following dialog gets opened up.

Still Not Available

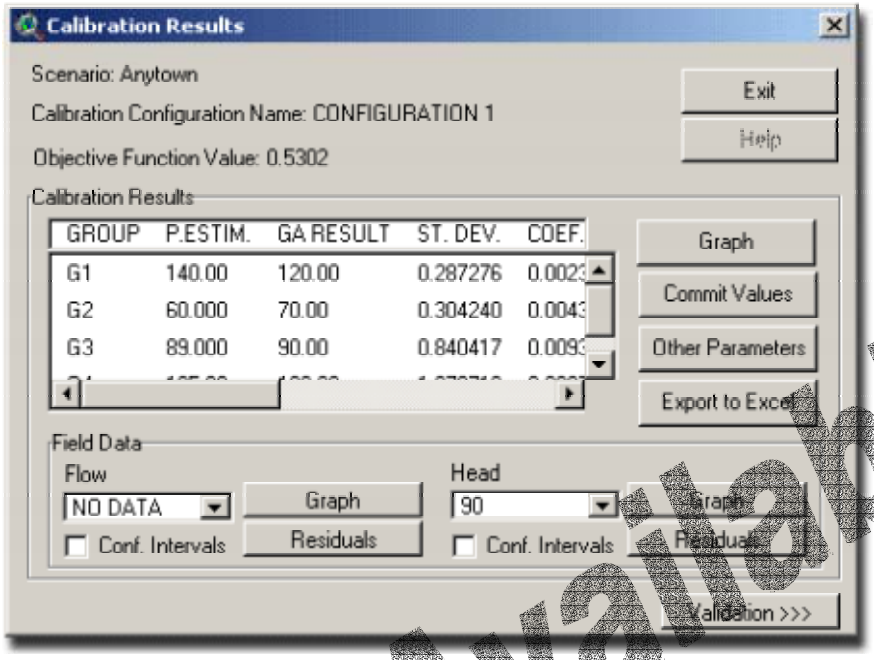
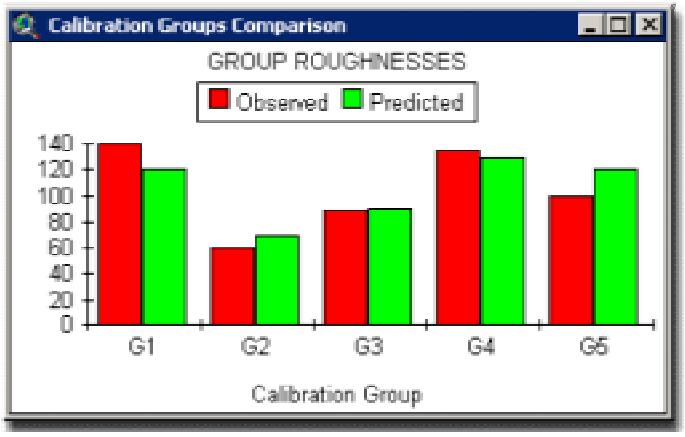


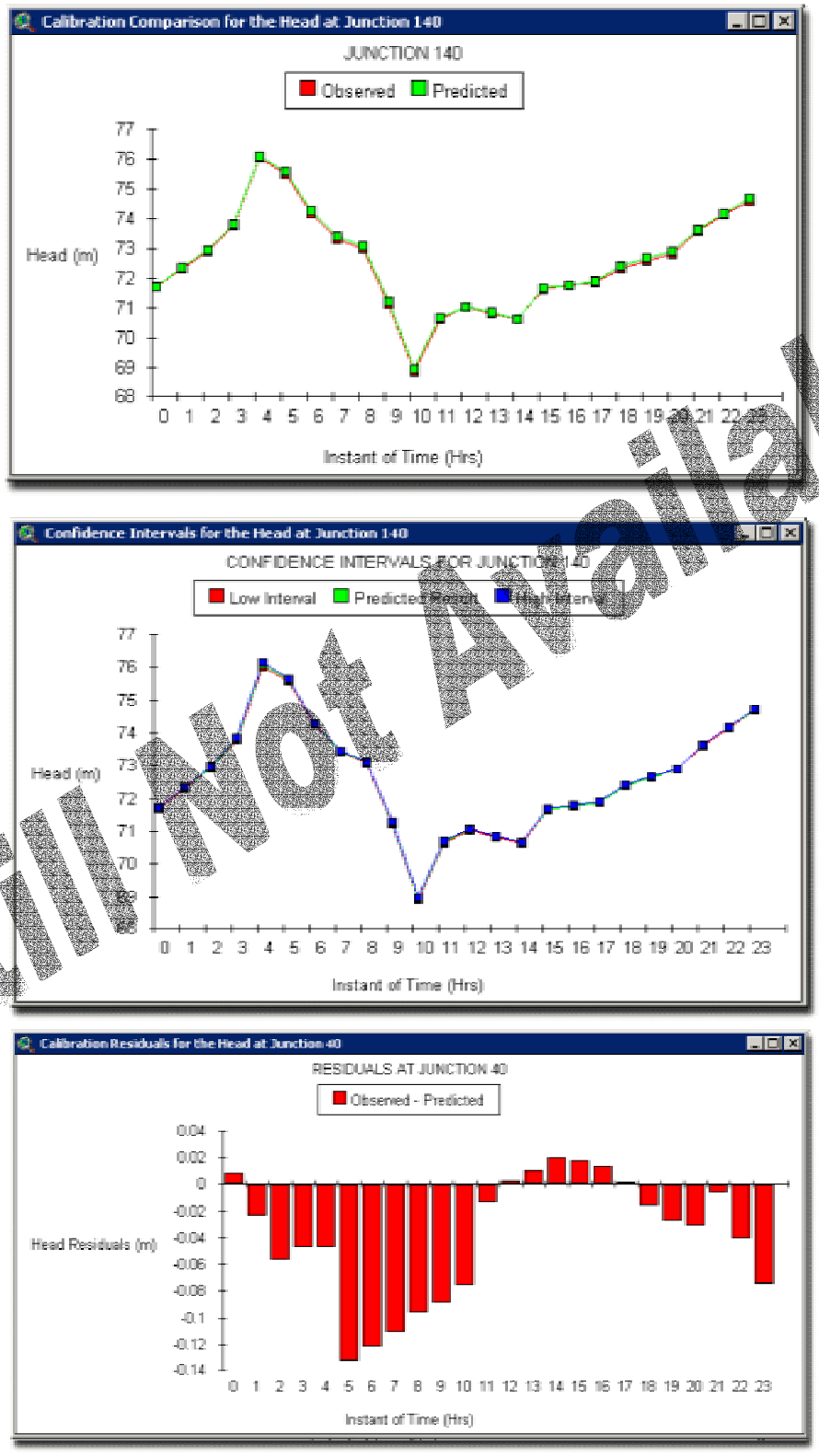
Fig. 91 Calibration Results Dialog.

This dialog allows to see several different kinds of graphs. All types are showed below. The first graphic shows the relation between the roughness of all pipe groups (use the **GRAPH** button beside the calibration results to see the graph). The two next graphics show the observed and predicted measurements (for each loading condition) without and with confidence intervals respectively (use the **GRAPH** button beside each one of the field data to see the corresponding graph, check the **Conf. Intervals** to see the confidence intervals along with the predicted values). The last graph depicts the residual values at the selected measurement location, namely the difference between observed measurements and predicted ones (use the **Residuals** button to see this graph).

Note: *In this example we are not going to use the Validation part as we used all measurements in the field data dialog to calibrate the network. It is customary for calibration purposes to leave several measurements out of the set just to use them later on for validating the calibration.*



Still not Available



Still Not Available

Fig. 92 Different types of graphs showing calibration results.

Another interesting thing that the calibrator computes is a whole set of useful regression statistics that can be used to assess the quality of the obtained calibration problem solution and the goodness of all those adjusted variables.

Use the button **Other Parameters** to see all those statistical analysis results. They can provide insight into the calibration process, quantify the quality of the particular water distribution system model calibration and identify data shortcomings and needs.

Finally, use the button **Commit Values** to replace the roughness of each pipe with those predicted roughness values calculated.

17.7.1 STATISTICS (Refer to Z. Kapelan thesis, 2002)

➢ **Decision Variables (Calibration Parameters) Results:**

- **ID**: Identifier of the decision variable.
- **Mean**: Predicted roughness value for the actual decision variable.
- **StDev**: Parameter standard deviation. Absolute uncertainty with which each parameter value is determined.
- **CoeffVariat**: Parameter coefficient of variation. Relative uncertainty with which each parameter value is determined.
- **95%LinearConfidenceInterval**: Linear parameter confidence interval is defined as the interval that contains the true value of the analysed parameter with a probability of 95%, *regardless* of the values of all other parameters.
- **CompScaledSens**: Composite Scaled Sensitivity. It is considered to be a relatively good indicator of the quantity of information provided by all observations (sample and prior information) for the estimation of a particular parameter. It is dimensionless and independent of the observed values, and therefore, the model fit. In an ideal case, all composite scaled sensitivities should have similar values, i.e. observed information should be equally distributed among parameters. According to Hill (1998), the ratio of largest to smallest css_j should not exceed app. 100 to maintain a well-posed problem. Obviously, parameters with large composite scaled sensitivities are candidates for splitting into two or more parameters (providing that they represent initially grouped parameters), while those with small values are candidates for either omitting from calibration or grouping with other parameters. Composite scaled sensitivities can also be used to indicate contribution of the potential new observation data.

➢ **OBJECTIVE FUNCTION**: The objective function is the fundamental statistic used to assess overall calibration model fit. *Weighted least squares objective function (see the note earlier on).*

- **Sample component of objective function value**
- **Prior estimate component of objective function value**
- **Maximum Likelihood Objective Function Value** :
  Another alternative to a weighted least squares objective is the maximum likelihood objective. By definition, likelihood *L* is the probability of occurrence of a specific instance of residuals *r*. Assuming independent and normally distributed residuals, likelihood is defined as follows (Carrera et al., 1986a):

$$L = \left[\det(2\pi C)\right]^{-\frac{1}{2}} \cdot e^{-\frac{1}{2} r^T C^{-1} r}$$

  where: *C* – generalised error covariance matrix (see above); *det()* - matrix determinant operator.

- **Root Mean Square Error**: RMSE $= \sqrt{\frac{1}{n} \cdot \left(\sum_{i=1}^{n} (y_i^* - y_i)^2 \right)}$, where: y* = observations, y = predictions, n = number of observations.

- **Number of Degrees of Freedom**: $\nu = N_s + N_p - N_a$, where N_s is the number of sampled observations, N_p is the number of prior estimates and N_a is the number of unknown calibration parameters.

- **Calculated Error Variance**: Defined as $s^2 = E/\nu$, where *s* is the standard regression error, *E* is the objective function value and ν is the number of degrees of freedom of the calibration model. If the fit achieved by regression is consistent with the observed data accuracy reflected in weighting W, the expected value of s^2 (and *s*) is one. Significant deviations of the calculated error variance from one indicate that the fit is not consistent with the weighting scheme. In practice, value of s^2 is, typically, larger than one, indicating usually the presence of model and measurement error(s) or larger than expected measurement error(s). If value of the s^2 is smaller than one it usually indicates existence of smaller than expected measurement errors. Obviously, in the general case when $W \neq I$, s^2 is a dimensionless number. In the special case when weights are not used (W=I, ordinary least squares), s^2 has same units as the measurement errors.

- **Calculated Standard Error**: ($\mu r/\sigma r$), where μr is the average weighted residual, σr is the standard deviation (spread of residual). It is used to indicate, in relative terms, how accurate the fit is, in an ideal case the value should be equal to zero (in practice usually less than a factor which is of the order of magnitude 10^{-1}).

- **Natural log. of the param. covariance matrix determ**.

- **Parameter Confidence Region Dimension**: Parameter confidence region addresses the probability that true values of two or more parameters are *simultaneously* within the specified region. In the case of two parameters, the joint confidence region is an ellipse whose actual size is defined by ellipse semi-axes which are proportional to the square roots of the corresponding eigenvalues of the *Cova* (parameter covariance matrix). The direction of the ellipse axis is defined by eigenvectors of *Cova*. The degree of the linear dependence (correlation) between parameters *a1* and *a2* is indicated by the angle between ellipse major axis and the horizontal (parameters *a1* and *a2* are independent when the mentioned angle is equal to zero). Eigenvalues and eigenvectors of the parameter covariance matrix can be computed by singular value decomposition of that matrix.

- **Trace of the Parameter Covariance Matrix**: Sum of diagonal elements of the covariance matrix. It is an indicator of the overall parameter uncertainty. **NOTE**: *Uncertainty* of an estimated model parameter is usually estimated by parameter variance which is estimated as the corresponding diagonal element of the parameter variance covariance matrix.

- **Condition Number of the Parameter Covariance Matrix**: defined as a ratio of the maximum and minimum eigenvalue of the above mentioned matrix (equal to 1 in an ideal case).

- **Correlation Coefficient R & R^2 (Cooley and Naff)**: Inspection of the model fit can also be achieved graphically by plotting weighted observations ($W^{1/2}y^*$) versus weighted model predicted variables ($W^{1/2}y$). Ideally, points should be distributed on the line with unit slope and zero intercept. A summary statistics which reflects how well the latter is achieved is given by the correlation coefficient (Cooley et al., 1990). In an ideal case, the value of *R* should be equal to one. According to Cooley and Naff (1990), values equal to or greater than approximately 0.90 indicate a good model fit.

Value R^2 can also be interpreted as a fraction of the total variance of model predictions that can be explained by the model.

- **Akaike Information Criterion (AIC) and Schwarz et al. Information Criterion (BIC)**: The statistics called information criteria (IC) accurately reflect the fact that as the number of calibration parameters increases, the reliability of parameter estimates (indicated by absolute or relative parameter uncertainty), on the average, decreases. Several IC exist nowadays. Two most widely used are **AIC** and **BIC** (Carrera et al., 1986b). They are defined as follows:

$$AIC = S(a) + 2.N_a$$
$$BIC = S(a) + N_a \,.\, \ln(N_o)$$

where: *S(a)* – negative natural logarithm of the likelihood. Obviously, the smaller the value of IC, the better the model. Note that above mentioned statistics are of indicative type only, i.e. in the case when values of AIC or BIC for a model with fewer parameters are only slightly larger than the statistics of another model, it is usually better to select the model with fewer parameters (unless additional reasons exist to select a model with more parameters).

For more details, refer to the Thesis of Z. Kapelan, February 2000. University of Exeter.

APPENDIX A. THE STANDARD GA LIBRARY

(Centre for Water Systems, University of Exeter, UK)

A.1 GENERAL GENETIC ALGORITHM CONCEPT

Genetic algorithms (GA) are one of the new evolutionary computing (EC) search techniques that have been developed in the past thirty years. These search techniques utilise many of the evolutionary processes in nature, to find near optimum solutions to real world problems. One of the most favourable of these EC search techniques is the GA. The term GA first appeared in the 60's, but their notoriety and popularity started in the seventies with the publications of Holland (1975) and De Jong (1975), and further with Goldberg (1989).

Essentially, a genetic algorithm optimises a given function as long as it can be explicitly defined. An example, that most people know is the following, find the value of $x \in (-1,1)$ that maximises the following function:

$$f(x) = 1 - x^2$$

GA mimics nature. They do this with the representation and the operators. As far as the representation is concerned, GAs use a coding to the problem similar to that of DNA's. A typical problem may be coded (using a binary representation) in a bit string as:

0 1 0 1 0 1 1 1 1

This value can be calculated, moving from the left to right as

$$X = \sum_{i=0}^{n} x_i * 2^i$$

where n is the number of bits that encode the problem, x_i is the bit value with i denoting the position of the bit where 0 signifies the left most bit, and n the right most.

This can be translated into a number (Y) between 0 and 1 using,

$$Y = \frac{\sum_{i=0}^{n} x_i * 2^i}{2^{n+1} - 1}$$

This can easily be scaled over any range. The value of Y is referred in the forms below as the search space.

Evaluating the function (f(x)) the objective value and sometimes the fitness itself. It should be noted though that often the fitnesses (with high penalty values) can cover a range in the order of several magnitudes. Often this means that fitness scaling function is applied. Therefore the terms, objective value and fitness refer to the function value and the scaled fitness respectively.

A GA begins by creating a population of randomly generated string bits. The fitness is assigned to each of these string bits. GA use a selection criteria that mimics natural selection - the more fit are more likely to be chosen to mate than the less fit. The chosen bits mate - exchange DNA information to create a new population. The exchange of DNA of the chosen

individuals is achieved through two operators; crossover and mutation. The simplest form of crossover is single point crossover, whereby the string bits are cut into two. The resulting children then is the first section of the first parent string bit and the second section of the second parent. Another child is made up of the remaining sections. Mutation is a random perturbation of one bit in the resulting children's bit string.

What happens next depends on the GA type. For the case of a generational GA the two child are added to the new population. New children are added until the new population is the same size as the old population, whence, the new population takes the place of the old and the GA continues by creating another population.

In the case of a steady state GA, the two children are checked whether they are fit enough to be put back into the population. This is determined by the replacer function. Once back in the population, two more individuals are chosen and mated.

In both GAs the process continues until convergence.

A.2 GENETIC ALGORITHM TYPES

A.2.1 Steady State GA

A constant population is used. The two individuals are selected and mated through the chosen crossover and mutation operators. The chosen replacer function determines whether the individuals are inserted back into the population. Therefore, the population remains constant, with every individual, even those just made, able to mate at any time.

A.2.1.1 Replacer Functions

Only the steady state GA requires a replacer function. It determines whether the newly made children are inserted into the population to continue through the generations.

- **Random**. The progeny replaces an individual at random.
- **By Rank**. Goes through the population to find the first individual that it is better than.
- **First Weaker**. Replaces the first weakest. It starts from the beginning of the population (the fittest individual) and iterates through to the last. The first individual that it finds that has a lower fitness value, it replaces.
- **Weakest**. The created individual is added to the population. The population is then sorted and the last individual is deleted (the least fit).

A.2.2 Generational GA

Differs from the steady state GA in that the whole new population is constructed each generation. This new population then replaces the old population. Another population is created in each subsequent generation. This GA has been shown to be slower in convergence.

A.3 SELECTION TECHNIQUES

The selection functions determines the individuals that are to undergo crossover and mutation. Points to remember in choosing the selection technique is:

- Trade-off between the diversity in the population and the convergence,
- The type of genetic algorithm to be implemented.

A.3.1 Uniform Random

The parents are chosen randomly from the population for reproduction.

A.3.2 Rank Biased

Rank biased selection uses the ranking of an individual to bias the selection. The GA library sorts the population from best individual to least fit individual. Placing a value to the Bias ($0 \leq Bias < 10$) places a bias toward those individuals with a highest fitness. As Bias $\Rightarrow$ 1 the higher the likelihood that a higher ranked individual is chosen.

$$Selection = Population.Count * \left(Bias - \sqrt{\frac{Bias^2 - 4*(Bias-1)*Random}{2*(Bias-1)}} \right) \quad \text{.....(1)}$$

For example, for a population count of 10, the table below gives the chosen individual given a certain random number (0.479743) and various *Bias* values.

Bias	0	0.2	0.4	0.6	0.8
Chosen Individual	7	7	6	6	5

Note: this equation falls over as does the whole GA if a bias of 1 is used.

A.3.3 Roulette Wheel

Each individual is assigned a probability of selection, or section of a roulette wheel, according to the ratio of its fitness over the total fitnesses of the entire population. This roulette wheel is then spun to determine the selected individual. The higher the individual's fitness the larger the proportion of the wheel it is assigned and greater the chance that it undertakes mating. This process can be represented mathematically as:

$$P_i = \frac{f_i}{\sum f_i} \quad \text{..... (2)}$$

where P_i is the probability that individual i is selected, f_i is the fitness of individual i. One of the weaknesses of this method is when the population (especially in later generations) consists of a number of individuals that have very similar fitness values. In this case, roulette wheel selection will not have a constant selection pressure, as the selection becomes a purely stochastic approach, unless a scaling function is utilised (Goldberg and Deb 1990).

A.3.4 Tournament

Tournament selection involves the random selection of a designated number of individuals from the population, who compete in a tournament for inclusion into the mating pool. Tournament selection, in its simplest form consists of a binary tournament involving the direct competition of two individuals for survival by the comparison of their fitness function, with the fitter of the two surviving. The tournament size can be increased, thereby increasing the selection pressure in the GA. This ranking of the population allows for a constant selection pressure (as in rank based fitness allocation). Use of tournament selection has shown better performance than the roulette wheel selection (Goldberg and Deb 1990). A variation of this selection mechanism, is the deterministic tournament selection, whereby the number of times that an individual can partake in a tournament in each generation is limited, thereby controlling the number of progeny each individual can have. Therefore, say in binary tournaments, the fittest individual in the population can expect two offspring in the next population, the average individual one and the least fit none.

- The tournament selection used in the GA library is a simple binary tournament, that randomly chooses the two individuals to go into the tournament from the population.
- Steady state GA limits the application to such a binary tournament.
- Tournament selection should not be used with a niching technique.
- Engelhardt (1999) used this for large tournament sizes, although this was with a deterministic type tournament selection using different code.

A.3.5 SRSWR (Stochastic Remainder Selection Without Replacement)

Works similar to roulette wheel selection. The ratio of each individual's fitness to the average fitness of the population is calculated. For each individual where this ratio is above one, a number of copies equal to the whole number of the ratio are placed in the chosen set. The remainder of the ratio, the decimal bit, is kept. The remaining individuals to make the size of the chosen set equal to that of the population are chosen from these remainder values using roulette wheel selection.

> **NOTE:** Currently, this selection scheme is only set up for the case of a steady state GA.

A.4 CROSSOVER TECHNIQUES

Crossover is the primary mechanism to share information from parents to individuals. Therefore, it is the fundamental mechanism that ensures that a GA performs better than a simple random search.
The major variable that controls this mechanism is the probability of crossover. This determines whether crossover occurs on the chosen individuals.

- Probability of crossover in range [0.5,1.0].
- As a guide use 0.9 for a steady state GA, and,
- 0.7 for a generational GA.

A.4.1 None

No crossover occurs. Fundamentally the GA thus becomes a simple hill climbing function, the hill climber being mutation.

A.4.2 Randomise

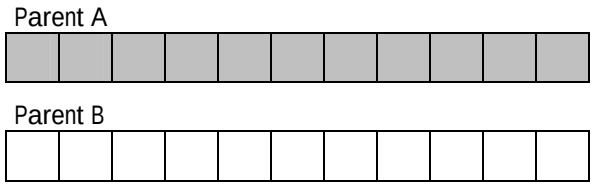
New random individuals are created every iteration. This is a true random search.

A.4.3 Simple One Point Crossover

A single location in the chromosome is chosen. The first child consists of all the genes located before this crossover point of the first parent, and the genes after this crossover point of the second parent.

A.4.4 Uniform Random

Uniform crossover (Syswerda 1989) differs form multi-crossover in the fact that each bit is considered as a section. A decision string, known as a mask is randomly generated deciding whether the bit should be placed into the first child or the second child. This is best shown in
.



(a) the parents

0 0 1 0 0 1 1 1 0 1 0

(b) the crossover mask

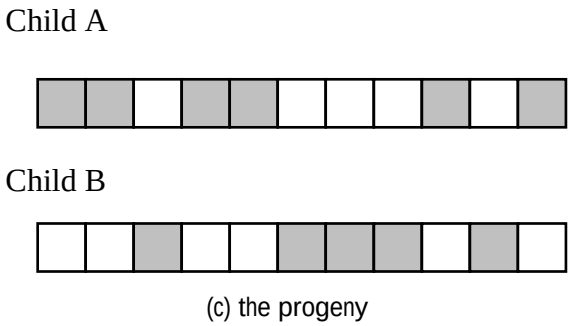
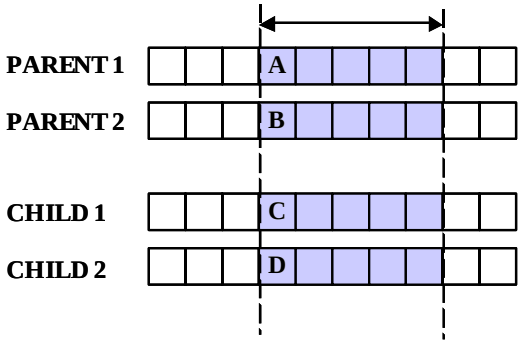


Figure 2: Uniform Crossover.

- Often will outperform one-point crossover, as there is no positional bias on the location that a decision variable is in the string as with one point crossover.
- There is a huge computational cost in using uniform random crossover.
- The exception to this rule if you are using bounded integer coding.

A.4.5 Arithmetic

This is a real type coding crossover operator. The child's gene is a randomly weighted sum of the respective genes in the parents.



w = random number $\in$ [0,1]

C = w.A + (1-w).B
D = (1-w).A + w.B

A.5 MUTATION TECHNIQUES

Mutation is a bit-wise perturbation operator. In other words, a bit within the chromosome is chosen at random and then randomly given a value. Mutation is controlled through the probability of mutation (P_{mut}) parameter. The higher the value, the greater the range of the solution space that one can expect the GA is to traverse, but this is at the expense of the GA converging quickly.
The literature, outside of some specific cases suggests that mutation should be no more than one bit a chromosome.

A.5.1 Uniform Random

The check on whether mutation is undertaken by checking the mutation probability against a randomly generated number. The coding then identifies a gene that can be mutated. A new value for this gene is created using a random number generator.

- The P_{mut} should be in the range of [0,1].
- A P_{mut} of 0 implies that no mutation occurs,
- A probability ≥ 1 implies that mutation occurs to one bit in the chromosome.

A.5.2 Uniform Random by Gene

Unlike uniform mutation, a integer value is calculated by multiplying the mutation probability times the chromosome length (ie.Random <= Parent.MutationRate * Chormosome.Length). This is then truncated (not rounded up) into an integer which implies the number of bits that are to be mutated. For this number, a bit is identified along the chromosome and mutated.
The value of the probability of mutation needs to be carefully thought out. If the probability of mutation is less than 1/(chromosome length) then no mutation will occur. If 1/(chromosome length) ≤ Pmut < 2/(chromosome length) then one mutation can be expected per chromosome. In effect, the choice of mutation rate implies how many bits will undergo mutation.

- A P_{mut} of 1 implies complete random mutation.

A.5.3 Non-Uniform Random

This differs from Uniform Random mutation in that the mutation is not random, rather follows heuristic rules implied by the programmer. Whether this heuristic rule is followed is user defined by the mutation bias variable. Essentially, as the GA converges, the *Allelle* value is either increased or decreased following a heuristic equation.

$$Allele = Allele \pm \left[\left(Allele - LowerorUpperBound \right) * \left(1 - Random * 1 - \%Complete^{Bias} \right) \right] \quad (3)$$

where, $$\%Complete = \frac{Iteration\ Number}{MutationMaximumIterations} \quad (4)$$

and *LowerBound* and *UpperBound* are the value applied as the respective bounds of the Allele value or the friction factors in the case that this mutator operator was designed for. The Bias and the MutationMaximumIterations are user defined factors.

A.6 CONVERGENCE CRITERIA

As GAs are stochastic in nature, a decision is required on when the GA has converged. There are two options available:

- **MIN. REQUIRED FITNESS VARIANCE**: The GA stops when the variance in the population reaches a given minimum that is placed in this textline.
- **MAX. ITERATIONS**: The GA stops when the total number of generations (or iterations for the steady state GA) reaches a given number.

A.7 OTHER VARIABLES

A.7.1 Population Size

This sets the number of individuals that make up the GA's population. The size of the population will have both an effect on the convergence and exploitation of the search space. A large population will mean that a larger proportion of the search space will be covered, but the longer the GA will take to converge.

A.7.2 Random Number Seed

GAs are a stochastic process, in other words all processes rely on the generation of a random number. Using the same random seed means that the initial population generated is the same. This allows any comparison undertaken in the genetic operators to start from the same position.

A.7.3 Output Interval

This is the number of iterations between outputting the best individual to date to an output file.

APPENDIX B – UNITS OF MEASUREMENT

Parameter	*US CUSTOMARY*	*SI METRIC*
Concentration	mg/L or ¤g/L	mg/L or ¤g/L
Demand	(see Flow units)	(see Flow units)
Diameter (Pipes)	inches	millimeters
Diameter (Tanks)	feet	meters
Efficiency	percent	percent
Elevation	feet	meters
Emitter Coefficient	flow units / $(psi)^{1/2}$	flow units / $(meters)^{1/2}$
Energy	kilowatt - hours	kilowatt - hours
Flow	CFS (cubic feet / sec) GPM (gallons / min) MGD (million gal / day) IMGD (Imperial MGD) AFD (acre-feet / day)	LPS (liters / sec) LPM (liters / min) MLD (megaliters / day) CMH (cubic meters / hr) CMD (cubic meters / day)
Friction Factor	unitless	unitless
Hydraulic Head	feet	meters
Length	feet	meters
Minor Loss Coeff.	unitless	unitless
Power	horsepower	kilowatts
Pressure	pounds per square inch	meters
Reaction Coeff. (Bulk)	1/day (1st-order)	1/day (1st-order)
Reaction Coeff. (Wall)	mass / L / day (0-order) ft / day (1st-order)	mass / L / day (0-order) meters / day (1st-order)
Roughness Coefficient	10^{-3} feet (Darcy-Weisbach), unitless otherwise	millimeters (Darcy-Weisbach), unitless otherwise
Source Mass Injection	mass / minute	mass / minute
Velocity	feet / second	meters / second
Volume	cubic feet	cubic meters
Water Age	hours	hours

REFERENCES

Bartolín, H. (1998). "Desarrollo de mejoras sobre una plataforma de enlace entre un Sistema de Información Geográfica (SIG) y un software de simulación hidráulica para la gestión de una red de abastecimiento. Aplicación a la red de Valencia". Proyecto Final de Carrera. ETSII, UPV.

Bartolín, H., Martínez, F., Monterde, N. (2001). "Connecting ArcView 3.2 to EPANET 2. A full environment to manage water distribution systems using models". Water software systems: theory and applications. International Conference on Computing and Control for the Water Industry (CCWI'01). Montfort University, Leicester, England.

Bosque Sendra, J. (1997). "Sistemas de Información Geográfica", 2ª Edición, Ediciones Rialp, S.A.

Christofides, N. (1975). "Graph Theory. An Algorithmic Approach". Academic Press.

Date, C.J. (1995). "An Introduction to Database Systems". 6th Edition, Addison-Wesley Publishing Company, Inc.

Elmasri, R., Navathe, S.B. (1997). "Fundamentals of Database Systems". Second Edition, Addison-Wesley Publishing Company, Inc.

Environmental Systems Research Institute, Inc. (ESRI, 1996). "Using Avenue". Redlands, (USA)

Environmental Systems Research Institute, Inc. (ESRI, 1996). "ArcView Spatial Analyst". Redlands, (USA)

Environmental Systems Research Institute, Inc. (ESRI, 1996). "Using ArcView GIS". Redlands, (USA)

Goldberg, D.E. (1989), "Genetic Algorithms in Search, Optimisation and Machine Learning", Addison-Wesley Publishing Co.

Hohl, Pat, Brad, M., (1997) "ArcView GIS Exercise Book", Second Edition. OnWord Press.

Hutchinson, S., Daniel, H., (1997). "INSIDE ArcView GIS", Second Edition. OnWord Press.

Huxhold, W.E., (1991). "An Introduction to Urban Geographic Information Systems". University of Wisconsin, Milwaukee (USA), Oxford University Press, Inc.

Kapelan, Z.S. (2002). "Calibration of Water Distribution System Hydraulic Models", PhD Thesis, School of Engineering and Computer Science, University of Exeter, United Kingdom.

Laurini, R., Thompson, D., (1992). "Fundamentals of Spatial Information Systems". Academic Press Ltd.

Macke, S. (2001). "DC Water Design Extension". Dorsch Consult. May 2001. http://dcwaterdesign.sourceforge.net

Macke, S., Battermann, A., (2001). "A Strategy to Reduce Technical Water Losses for Intermittent Water Supply Systems". Thesis

Martínez, F., García Gómez, C. (1998). "Integración del programa Epanet para el análisis de redes de distribución de agua en ArcView 3.0" VII Conferencia Nacional de usuarios ESRI. Madrid.

NEOS (1996), “Server for optimization”. http://www-neos.mcs.anl.gov/neos/

Razavi, A.H., (1997). “ArcView GIS/ Avenue Developer’s Guide”, Second Edition. OnWord Press.

Razavi, A.H., Warwick, V., (1997). “ArcView GIS/ Avenue Programmer’s Reference”, Second Edition. OnWord Press.

Rossman, L. (2000), “Epanet 2 User’s Manual”. Environmental Protection Agency. Cincinnati, USA. (http://www.epa.gov/ORD/NRMRL/wswrd/epanet.html)

White, C. (2001). “Using GIS to implement hierarchical organization in water distribution system simulation models”. 20th Annual ESRI Conference, San Diego.